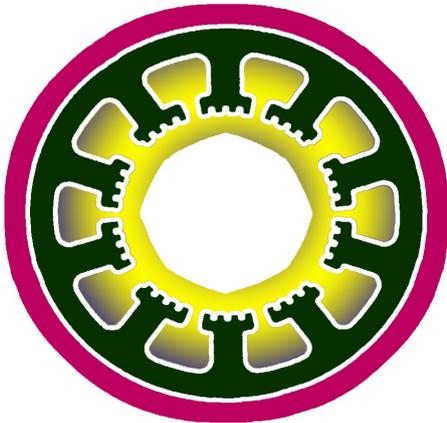**milling, plotting, engraving, drilling, grinding, dispensing, cutting, and much more with...**

WinPC-NC

Professional and USB

Software that turns your personal computer into a universal stepper motor controller

Version 4.00

March 2021

Legal Notice

Lawful purchase of the *WinPC-NC* control program conveys permission for one person to utilize the *WinPC-NC* control program and its manual. Copying or modifying the *WinPC-NC* control program - including any of its constituent parts - or its manual are forbidden. Furthermore, any unauthorized transmission of the program or its manual - including any extracts - will be legally prosecuted by all means available.

The authors reserve all rights to the *WinPC-NC* program, its constituent parts, and its manual, including all accepted forms of copyright.

WinPC-NC may not be used in association with medical equipment or food preparation without the explicit, written permission of the author.

Despite extremely thorough testing, it is never possible completely to eliminate all faults and ensure problem free operation. No responsibility can be accepted for damage caused as a result of using our program. We would be grateful to receive feedback, positive and negative, and detailed reports of any problems from lawful *WinPC-NC* users.

Support and reduced cost updates are available for registered customers. Registration is performed by sending us an email containing the *WinPC-NC* version number, your assigned serial number, and your mailing address to the email address below. This information may be found on the installation media or visible in the program window when *WinPC-NC* begins execution.

Please register your license!!!!

Content

About this manual

The *WinPC-NC* manual is divided into chapters:

Chapter 1 :   The basics: computer requirements and potential uses.

Chapter 2 :   Quick start guide: installation, initial program set up, and use..

Chapter 3 :   Description of all program features.

Chapter 4 :   2D-CAM functions: Sorting and tool diameter compensation.

Chapter 5 :   Parameter and option descriptions.

Chapter 6 :   Detailed, step by step start-up procedure.

Chapter 7 :   Signal Wizard: How input and output signals are defined and assigned.

Chapter 8 :   NC data format interpreters and error messages.

Definitions

Some terms used in this description must be speficied:

Job or NC file   A file with NC data which is read and processed by *WinPC-NC*. The file may contain many types of data - milling, plotting, engraving, drilling, and grinding

Job or process   The reading of an NC file and its resultant effects on the attached NC machine.

Command   An individual instruction in the job file which gives rise to one or more actions on the attached NC machine or in *WinPC-NC*.

Button   Clickable field that performs a specific function.

Checkbox   Clickable box which activates or deactivates a specific parameter or function (e. g. signals). An activated checkbox is marked with a cross.

Axis assignment depends on how you view your machine:
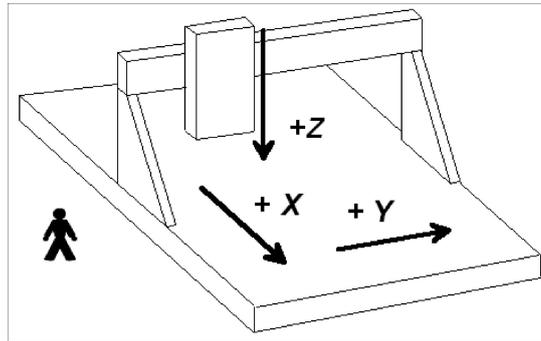
Axis assignments with your view parallel to the gantry
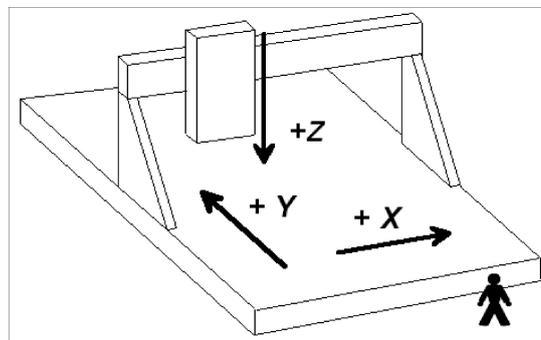
Axis assignments with your view perpendicular to the gantry

Use of typography

Keyboard action	Normal script enclosed in a box: ENTER
Cursor keys	Normal script enclosed in a box: UP LEFT PGDN
Menu names	Hyphen separated words, all in upper case: **FILE-DISPLAY**
Messages	Italic script: '*Perform reference move ment?* Y/N'
Function names	Capital letters: **SIGNAL TEST**

Different *WinPC-NC* Versions

WinPC-NC Light is our low-cost offering, perfect for beginners. One can engrave, mill, drill, and create PCB boards and perform 2.5D milling. Machine control is handled via *nc100*, an external USB adapter capable of controlling one IEEE 1284 compatible port (LPT).

WinPC-NC USB is an enhanced version of *WinPC-NC Light* that adds additional NC data format interpreters and many useful features. Operations may be performed in 2.5D, as well as with 3 and 4 axes. Machine control is handled via a choice of external USB module:

nc100   Supports one IEEE 1284 compatible port (LPT).

ncUSB   Supports two IEEE 1284 compatible ports (LPT).

WinPC-NC Professional is our version designed for industrial use. Machine control is handled via *CNCCONS*, our external motion controller. The *CNCCONS* controller offloads timing and CPU intensive operations from the PC. True real-time control with the utmost stability and reliability is possible. Custom hardware configuration, software, and mounting services are available.

The system is quite independent of windows speed and provides besides utmost stability, reliability and additional professional technology functions.

Orientated to requirements of individual applications, signal support and various casing types,suitable for being integrated into control cabinets, are available.

The axis controller is equipped with a documented API program interface and thus own and specific host programs can be used.

The different versions can be compared by an extensive function table document and provides good overview concerning operational performance and specific options.

Please contact us (info@lewetz.de) if there are special requirements (e.g. direct LPT control, SMC stepper cards, custom motion controllers) for your aplication.

Extended functions in *WinPC-NC Professional*

Following options can solely be used by *WinPC-NC Professional*…

- Dosing/dispensing technology with nozzle to prevent dripping

- Flame and plasma cutting technology with height correction and Z zero sensor system

- Optional extension up to 265 inputs/outputs with 24V signal levels, suitable for being integrated in control cabinets and analogue output of 0-10 V.

- Monitoring of inputs and freely definable message display

- Tool changer with 4th axis, e.g. as drawer or circular magazine

- Digitalization of non-planar surface for reproduction

- Double and multiple heads with several Z-axes and automatic changeover

- External job selection with inputs

- Communication and synchronization with SPS and other controllers

- Saving the complete job on the axis controller and thus mass production without the necessity to connect a computer

- Display of signal-test can be personalized for used and monitored inputs/outputs

- Documented API program interface for control process with own host programs.

- Double X-axis with two motors and automatic rectangular alignment of gantry at homing

- Various options for remote control by separate and superior Windows host programs for individual data transfer

- Various operating data counters for spindle and machine run time or grooves

- Axis controller with integrated SPS runtime system to ensure simple and efficient switching and monitoring functions

- Individual OEM versions for complex and very specific hardware and software requirements

1. What can *WinPC-NC* do?

Universal program

WinPC-NC is a software program which takes any standard personal computer and turns it into a universal NC control system for up to 4 axes.

Required hardware accessories encloed

WinPC-NC USB uses a small USB module called ncUSB. Thus CNC machines or drives can be directly controlled. The USB module is equipped with 2 LPT compatible ports for connecting the machine and using additional inputs and outputs. Optionally another type of USB hardware can be used, which is already integrated.

WinPC-NC requires a modern computer with the latest technology, i.e. 2 Ghz clock pulse rate is recommended as well as a 32-Bit or 64-Bit operationg system starting from WinXP to Win10 (status 01/2021).

ncUSB box and cable

Individual and user-defined 3D mechanical structures can be realized by four stepper motor axes and thus multiple tasks can be managed.

Applications include:

• **Milling**	• **Plotting**	• **Tangential cutting**
• **Drilling**	• **Welding**	• **Foam cutting**
• **Grinding**	• **Laser cutting**	• **Foil cutting**

- **Engraving**   - **Plasma cutting**   - **PCB prototyping**

- **3D Printing**   - **3D Scanning**   - **Dispensing**

- **Fabric and plastic
  sheet cutting**

WinPC-NC Professional comes with a more powerful axes controller cpu and controls the CNC machine independent from Windows services and background tasks much more reliably and stably. It is our recommended solution for hard and industrial tasks and applications.

Axes controller in standard housing for *WinPC-NC Professional*

Variety of parameters	Due to the variety of parameter modes the program can be adapted to nearly all 2-4 axis machines.
Cleary structured operator interface	*WinPC-NC* provides a clever and modern operational concept with drop down menu, Window management system and mouse operation. It is easy to learn and to manage.
Runs on any modern PC	Using *WinPC-NC* means a modern standard PC with hard disk, USB port, any graphics card and a Windows operating system. Notebooks and netbooks with appropriate technical features can also be used.
	The axis controller of *WinPC-NC* is connected to the system via RS232 port or by means of a standard USB-RS232 adapter .

example for a 3 axis milling machine

Special features

Special features of *WinPC-NC USB* and *Professional* :

- Offers a well thought out user interface that is easy to learn. A context sensitive help system makes getting work done easy.

- Offers an almost unlimited working range

- Handles stepping frequencies up to 80 Khz (interface dependent) with 4 axes synchronously (4 x 80kHz).

- Uses keyboard or mouse commands precisely to jog (move) around.

- Displays your NC data graphically. Scale, zoom, shift, rotate, and reflect as necessary.

- Allows axis resolutions, speeds, backlash compensation, and ramp length for the X, Y, and Z axes to be set within generous limits.Allows axis resolutions, speeds, backlash compensa-

tion, and ramp length for the X, Y, and Z axes to be set within generous limits.

- Supports a 4th axis. It can be programmed as U, V, W, A, B, C, or tangential axis T

- Supports home and limit switches: up to 10 inputs and 8 additional outputs can be set within generous limits

- Outputs signals that can be used for synchronizing with external equipment (e.g. job start, spindle speed reached, machine ready)

- Recognizes HPGL, DIN/ISO, MultiCAM 2D and 3D, drilling formats, ISEL NCP, EPS/AI and DXF (2D).

- Has extensive tool management functions. Tools can be selected individually, colors can be assigned; feeds and speeds specified; unique characteristics handled.

- Supports automatic tool changers. Up to 10 magazine positions are supported.

- Has CAM functions for cleaning up drawings. Sorting by tool and position; tool diameter compensation.

- Handles speed settings as slow as 0.01 mm/s and as fast as 2000 mm/s.

- Offers an integrated high-performance file editor. An external editor may be specified and used as well.

- Supports speed control of drilling/milling spindles. Rotation can be clockwise or counterclockwise.

- Supports Metric and Imperial units. Dimensions may be millimeters or inches; speeds may be mm/sec, mm/min, or inches/min

- Works with multiple languages: 22 languages are available at the time of writing; more are being added.

- Has a flexible macro language. Functions can be called at various points in a job.

- Allows feed and spindle speeds to be overridden at any time.

- Supports automatic and manual tool length compensation.

- Has Manual Data Input (MDI) capability. Enter low level commands.
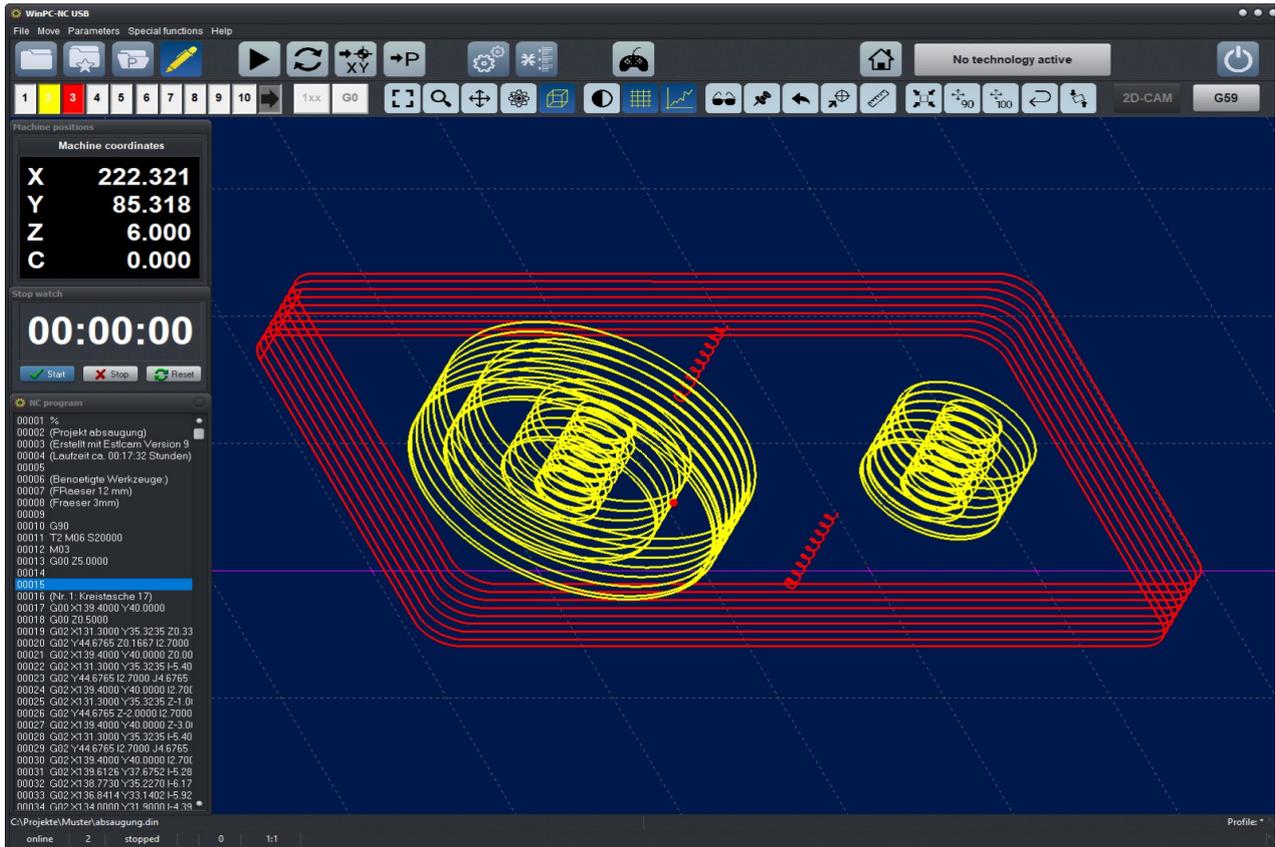
The *WinPC-NC* user interface

- comfortable teachin function including arcs and splines

- remote control via Windows registry or external input signals

- supports pendant, hand wheels and keypads

- separate settings for machine and mechanic and job specific parameters

- profile management for quick selection of different applications

- Supports job nesting

- Power control for lasers with PWM-Signal

- ...and much more besides

2. First Steps

2.1. Requirements

- Personal Computer (PC) - desktop, laptop/notebook, or net-book - running at a minimum of 2 Ghz clock speed.

- 50 MB of free storage space.

- 2 GB of RAM. 4 GB or more is strongly recommended. Memory requirements grow with the size of the NC file being processed.

- Screen resolution of 1024x768. A higher screen resolution is strongly recommended as this allows for increased flexibility.

- A keyboard and a 2 or 3 button mouse. A scroll wheel is strongly recommended.

- 32-bit or 64-bit version of MS-Windows - Windows 7 or newer.

- The OS and all its drivers should be up to date.

- Deactivation of any CPU and system power saving settings, screen savers, hard disk indexing and defragmentation programs, automatic update checking, automatic shutdown capabilities, and any CPU clock speed adjustment features.

- A free USB 2 port for an external USB module (e.g. **nc100, ncUSB**)**.** The power saving settings for USB ports and device manager must be deactivated. Do not use a USB hub or switch between the PC USB port and an external USB module.

- When using **nc100** or **ncUSB**, be sure to use the bundled USB cable.

WinPC-NC USB is limited to working with the **nc100** and **ncUSB** modules. You cannot use any 3rd party USB hardware to control your machine with **WinPC-NC**.

WinPC-NC USB ensures that correct USB driver is available for operation with the **nc100** and **ncUSB** USB modules**.**

WinPC-NC Professional controls the included axes controller via a serial RS232 link or a USB-RS232 adapter. Therefore a serial COM port is needed.

2.2. Installation

User-friendly installation

WinPC-NC is installed using a user-friendly installer program. Insert the CD into your CD/DVD drive and wait for automatic installation to start. If the installation does not start automatically, open the CD drive and start the installation by double clicking SETUP.EXE.

The installation wizard guides you through the installation procedure.

When *WinPC-NC* is first being installed, do not connect an external USB module (e.g. *nc100*, *ncUSB*) until after completing the installation procedure.

If there is no CD/DVD drive available in the control computer, the SETUP.EXE file can be copied from the CD to another computer or USB stick and installation can be executed in this way. Access to the setup file can also be made by network. The control computer and *WinPC-NC* do not need network access.

File list

When installation completed, the target directory will contain the following files:

WINPCNC.EXE	The *WinPC-NC* program
WINPCNC.WPI or WTI	Parameter file
WINPCNC.WPW or WTW	Machine tool file
WINPCNC.WPO	CAM function settings
WINPCNC.MAC	Initial or user defined macros
WINPCNC.PDF	This manual in PDF format
WINPCNC.LNG	Multilingual messages and texts
WINPCNC.LIC	Personalized license file
README	Important, late breaking information
*.PDF	Various additional information
*.DLL	Files required by the system folder
Folder PDF	Online-Help-texts
Folder PROFIL and TEMPLATE	Templates and data for profile management
*.PLT *.SMM *.DIN *.NCP *.EPS *.DXF *.TAP *.NC *.GC *.CNC	Sample NC files

Be sure to open and read the README file as it will contain important information.

Don't plug in ncUSB until software installation is complete

After the **WinPC-NC** installation completes, plug in your external control module. For the **nc100** and **ncUSB**, choose any available USB2 port.

Connecting an **nc100** or **ncUSB** module will result in an immediate hardware identification message followed by driver installation. When driver installation completes, you will receive message that the module is ready for use.

2.3.   Launching the program

WinPC-NC is launched simply by clicking the desktop icon or from of the start menu.

If **WinPC-NC** came bundled with your CNC machine, the first time the program starts, a menu will appear asking you to select your machine. Your selection will ensure that the machine parameters and settings appropriate to your machine will be used.

The **WinPC-NC** working window will appear.

Different views

The view of the main screen can be customized and personalized by individual settings and, in fact, used with a touch optimized function.

Detailed information on different views as follows:

2.3.1. Standard layout
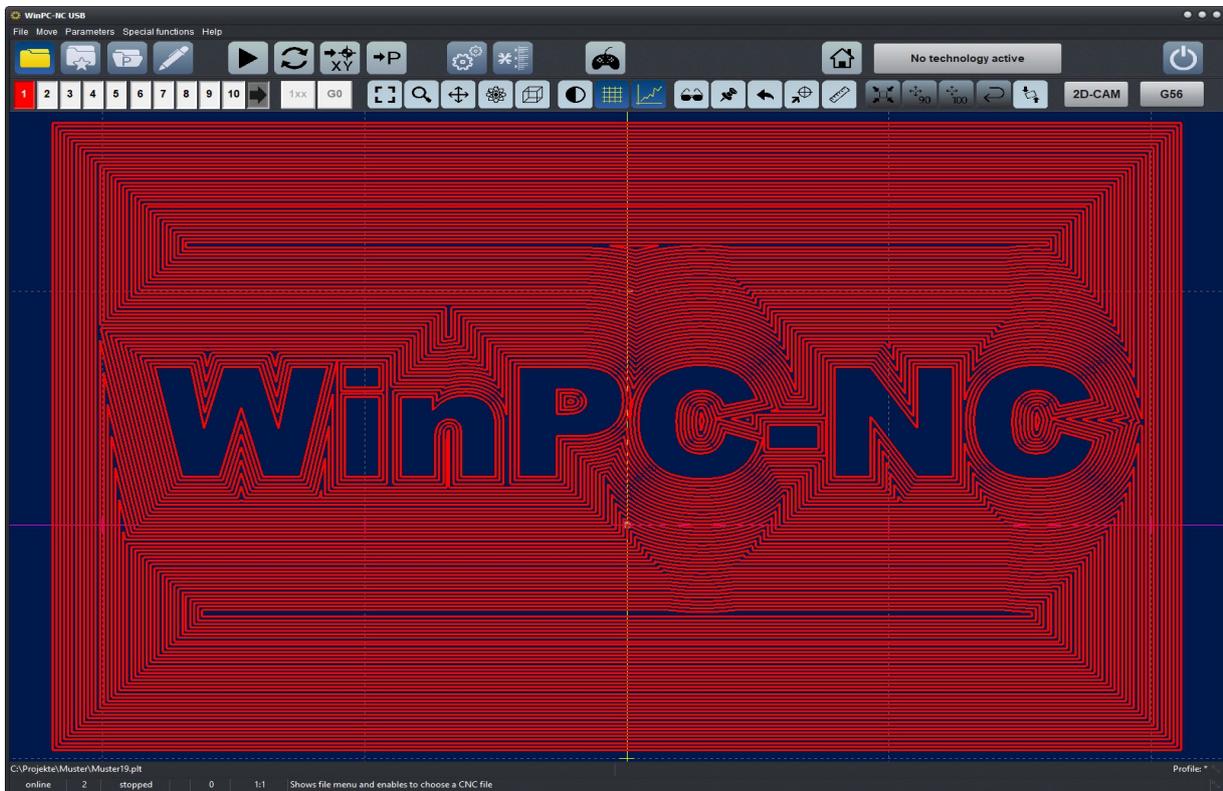
WinPC-NC main window, the default layout after the installation is the standard layout

Screen layout   Title bar and menu bar are placed at the top corner of the screen from where the drop down menus are opened.

- **Title bar -** The *WinPC-NC* working window title bar.

- **Menu bar -** Drop-down menus open from the menu bar.

- **Button bar -** Useful buttons for rapidly performing many actions.

- **Tool bar -** Buttons for NC data display and manipulation.

- **Display Area**

- **Status bar** - Multipurpose status display area:
  The *nc100/ncUSB* status (online/offline).
  The operating status of the software.
  The operating status of the NC machine.
  The emergency stop status (in red).
  Error and warning messages.
  Helpful texts concerning the action being performed.

The button bar icons:

Icon	Description	Icon	Description
	End the program		Joystick movement
	Open an NC file		Initialize the machine
	Load current or new file in the editor		Parameter settings
	Call profile management		Handling of molette
	Start Job		Tools
	Jog		Show/Open last opened NC files
	Move to current zero point (X/Y)		Move to current park position
	OK - Yes		Cancel
	No		

Buttons concerning display and tools are located in the second button bar.

At the bottom line of the screen you will find any information concerning operating status of the software and the machine or a help text about the functions which are currently active is displayed.

The status of the emergency stop is also signalled in this area and possibly displayed with a red field. Furthermore, it is possible to read off the current cursor position in machine and workpiece coordinates as well as the current width of the grid pattern.

The large area is the working area where *WinPC-NC* displays the current loaded NC file in graphical format.

Windows can be freely moved and arranged

Switchable additional windows, e. g. for axis position, program code and stopwatch can be freely moved and organized. These positions are durably stored in *WinPC-NC.*

2.3.2. Layout with Side-Panel

Rapid machine operation

For an easy and comfortable use of setup functions a permanently visible side panel can be displayed which constantly shows travel functions for the machine setup and thus provides an uncomplicated handling.
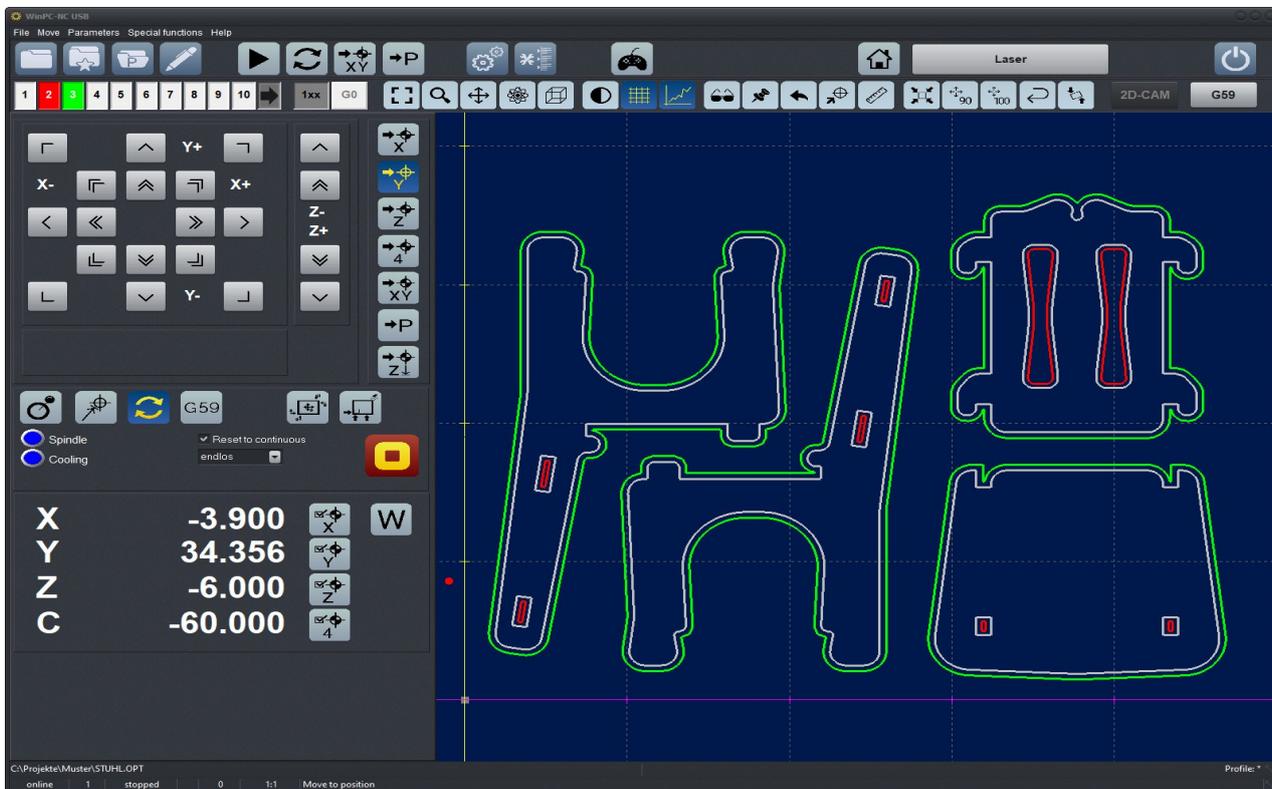
WinPC-NC main window, with Side-Panel

After starting the job the side-panel view changes to the job view and allows override operation by displaying the current loaded NC program together with the actual processed program line.

After job end or abort the travel functions are immediately visible again and thus it is easy to move or park the tool.
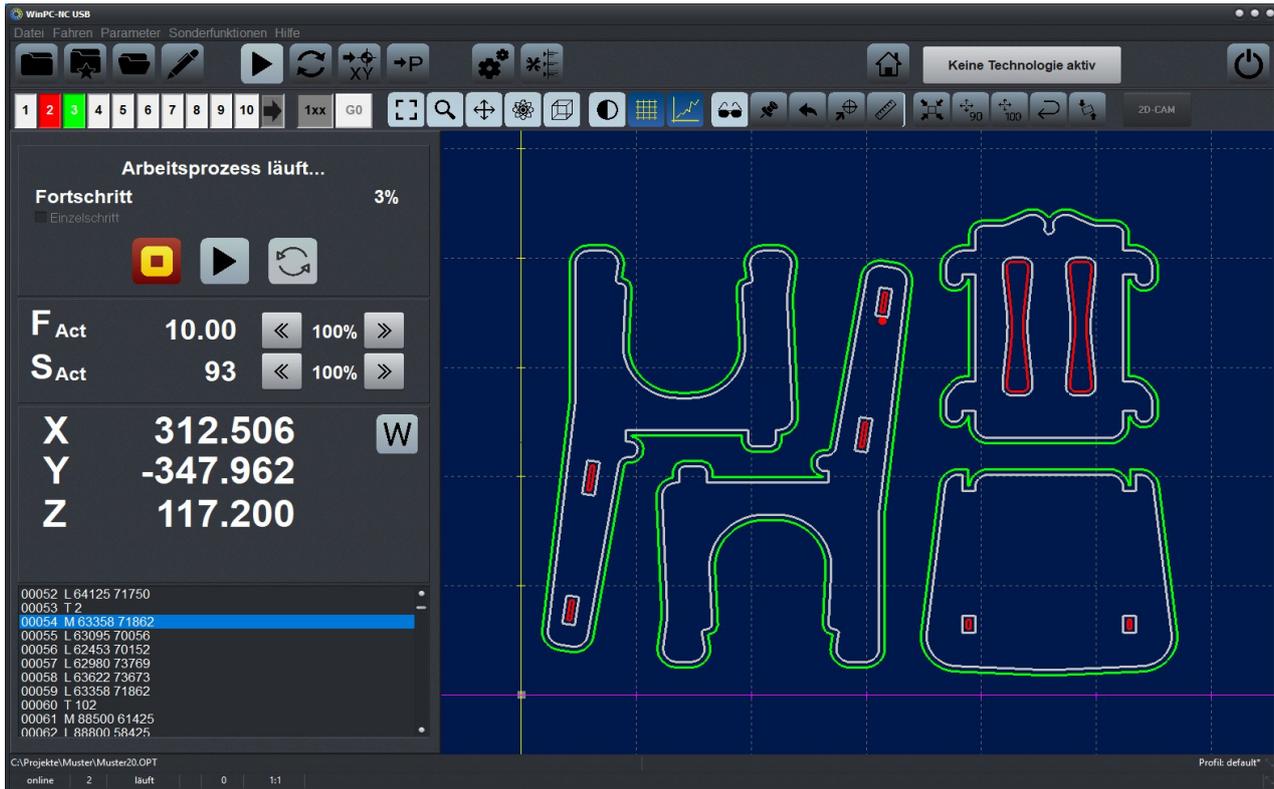
WinPC-NC main window, with Side-Panel - when job is running

2.3.3. Touch optimised layout

Control buttons and display panels are conceived much bigger for easy operation without mouse and they are optimized for touch-sensitive displays.

Due to an improved and enlarged design of all callable dialogs and functions, comfortable handling without the risk of being mixed up is guaranteed.

The touch view can be displayed as an overview with side-panel or as enlarged graphics view. The switchover is made by both buttons – *OVERVIEW* and *GRAPHICS.*

Display - Overview
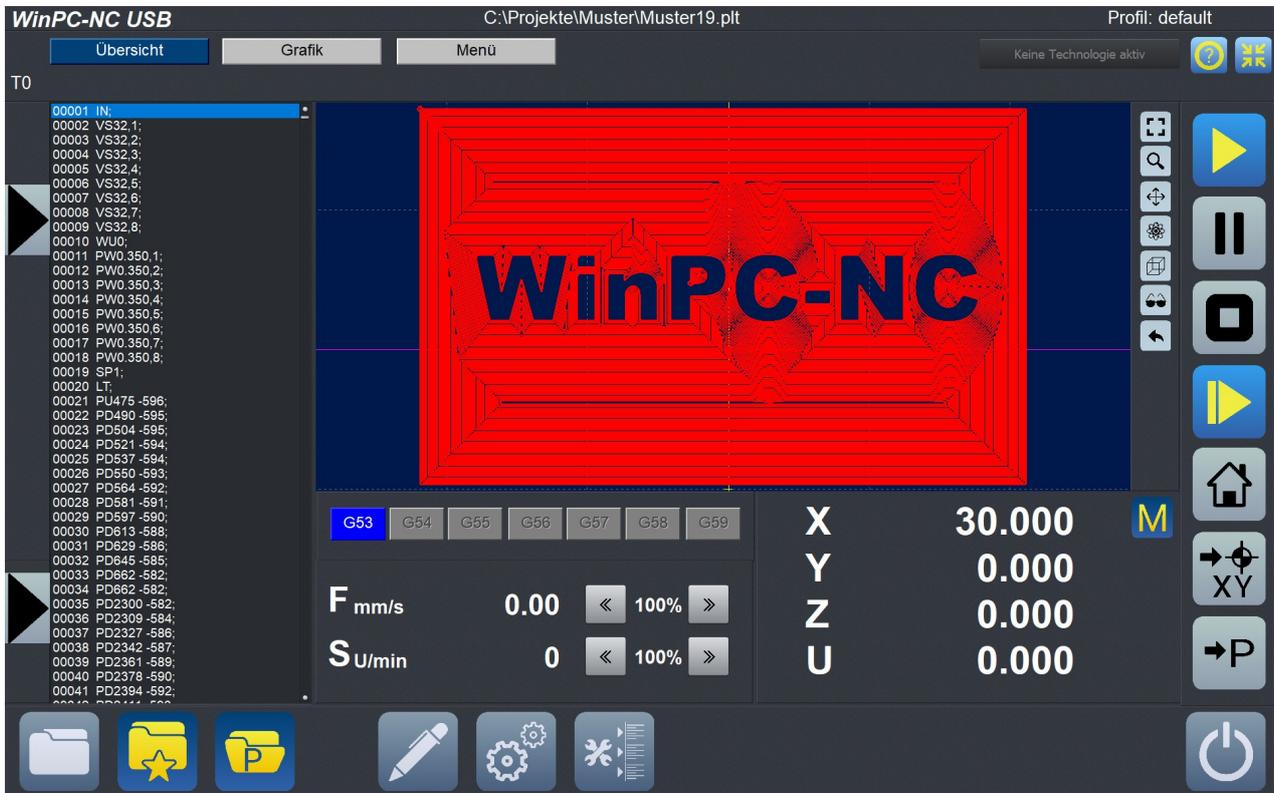
WinPC-NC main window - Touch layout

Screen layout   Title bar and 3 buttons for selecting specific modes are placed at the top of the screen.

Buttons which are relevant for administrative tasks in **WinPC-NC** are placed at the bottom of the screen.

Moving functions can be executed by buttons placed in the right line.

On the left side the NC window is firmly embedded (overview).

Below, the area is subdivided into a program window of the loaded file, graphics view, current machine data, e g. positions, speeds and buttons for calling up required functions.

From the left edge a dialog can be displayed enabling manual traverse of the axes and the setting markers.

Title bar

The name of the loaded file is displayed in the middle of the upper area.
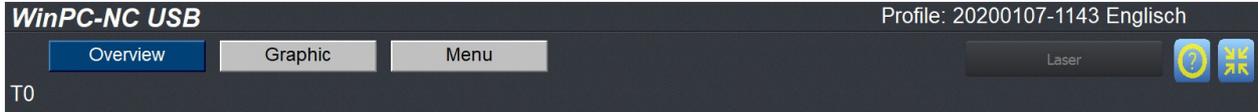
At the right side the loaded profile. If the profile has been changed, it is marked by * at the end of the profile name.

Overview - **Graphics** – **Menu buttons**. The various modes can be selected by the three buttons.

The **technology button** signals a possibly active technology. Pressing this button means direct access to the parameter page of the technology.

The current selected tool is displayed on the bottom of the left corner in the title bar.

The buttons:

	Quit *WinPC-NC*		Joystick move-ment
	Open an NC file		Initialize the ma-chine
	Load current or new file in the editor		Parameter set-tings
	Call profile management		Handling of mo-lett
	Start or continue Job		Tools

Pause job (only if „enable job resume" is checked in parameter settings)

Stop job

Job start from

Jog

Show/Open last opened NC files
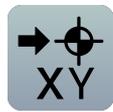
Move to current zero point (X/Y)

Move to current park position

OK - Yes

Cancel

No

Show help

Minimize

WinPC-NC

The display of the machine / workpiece coordinates
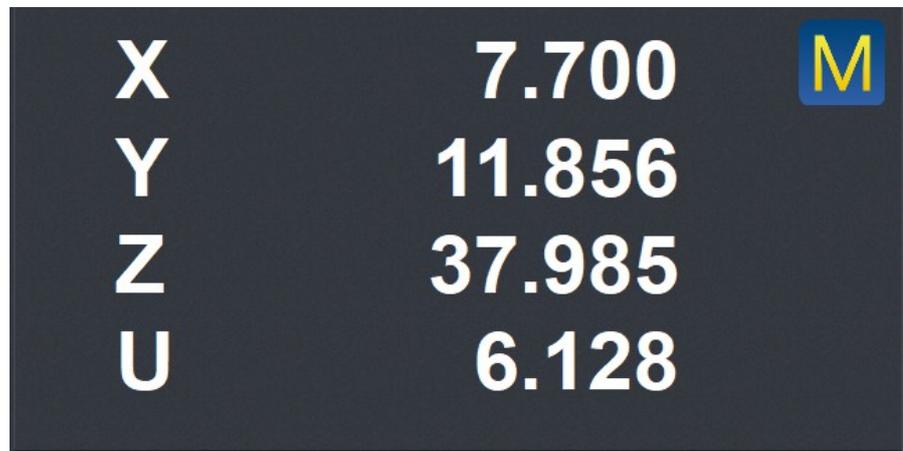
With the button at the top right you can change the display between the machine / workpiece coordinates.

The display of the job plunge and the current spindle speed

F signals the current feed rate. Speed can be increased or reduced during the job runs via arrow keys.

S signals the current spindle speed. Speed can be increased or reduced during the job runs via arrow keys.

The activce zero point is marked in (one of) the seven boxes above and can be easily changed.

The stopwatch

The stopwatch is displayed at the right side at the bottom of the screen after the relevant parameter has been activated.

Opening the manual window is made by the bar at the left side of the screen. By moving the cursor on the area with the arrow keys, the manual window opens.

The window will be closed again as soon as the cursor leaves the area of the manual window.
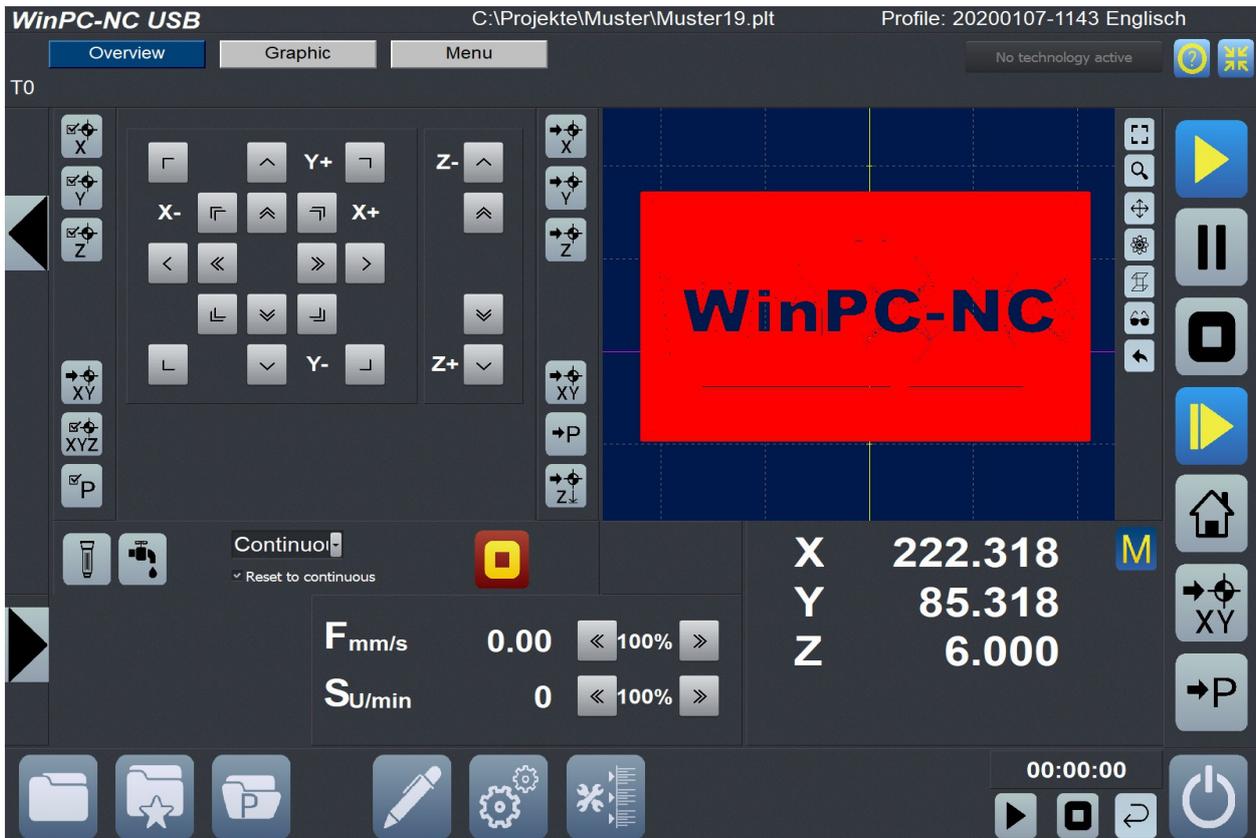
Unlocked opened jog window

Opening the manual window is made by upper arrow key on the left side of the screen

When you press this arrow key the manual window "*locked*" opens, i. e. the window is closed again after reactivating the upper arrow key. The lock symbol is displayed at the right side of thetop of the screen.

Locked opened jog window

Opening the line for selecting manual functions is made by down arrow key on the left side of the screen.

By pressing the down arrow key the line for selecting Jog move opens. Closing the line is also made by this arrow key.
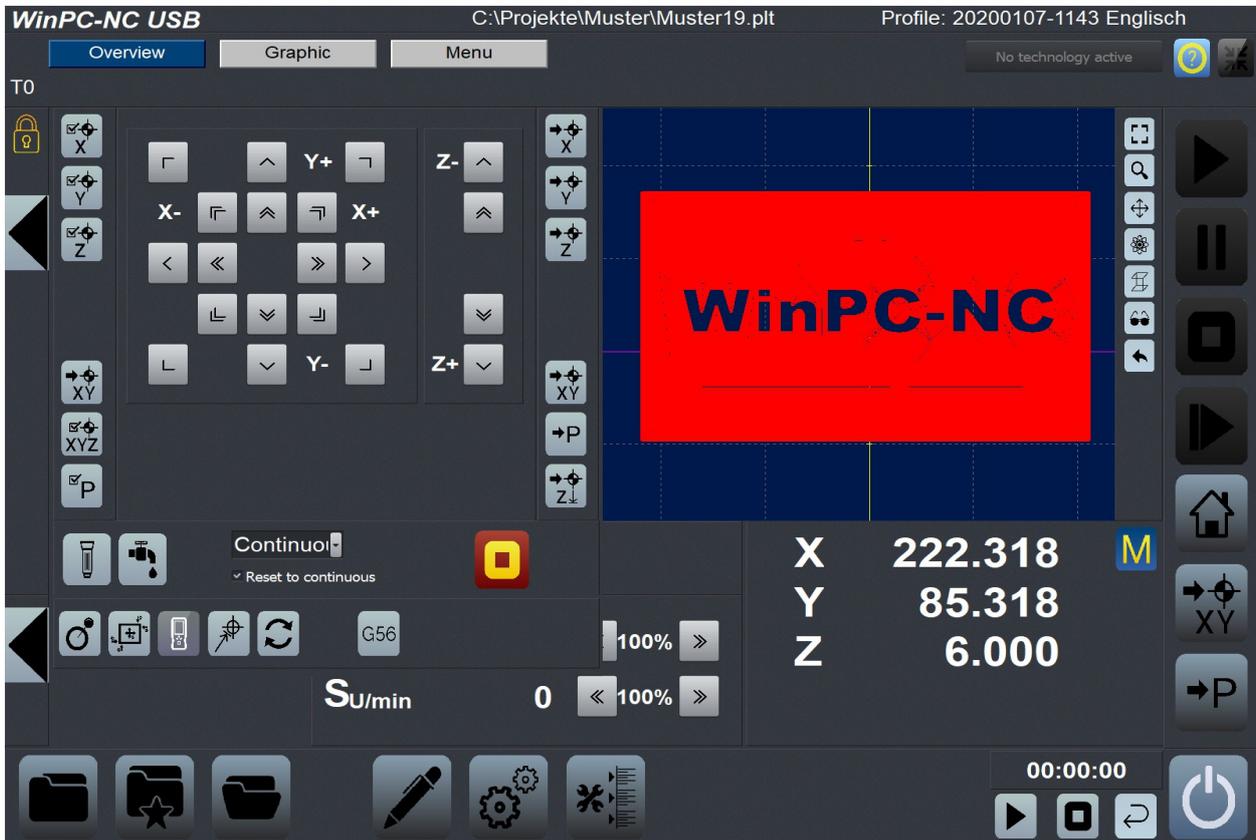
Locked opened jog window with opened jog functions bar

Overview - Possible graphic functions :
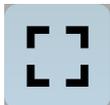
Reset display view

Restores the Display Area to its initial condition. Any changes to the NC data will be lost. If a mode is selected, it will be deselected.

Zoom the data

The NC data may be zoomed - *magnified* - by activating this mode. The mouse cursor will change to notify you that zooming is in effect. Click the left mouse button in the *Display Area* and moving the mouse around will cause the NC data to zoom.

Zooming can also be done by moving the mouse scroll.

The NC data may be scrolled – *moved around* - by activating this mode. The mouse cursor will change to notify you that scrolling is in effect. Click the left mouse button in the *Display Area* and moving the mouse around will cause the NC data to scroll.

Scrolling can also be done by moving the mouse around while in the *Display Area* and keeping the right mouse button pressed.
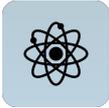
Orbit the data

The NC data may be orbited - *rotated around* - by activating this mode. The mouse cursor will change to notify you that orbiting is in effect. Click the left mouse click in the *Display Area* and moving the mouse around will cause the NC data to be orbited.
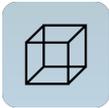
Standard perspective views

Clicking this button cycles the display through 4 useful perspectives.

Select the display area

Display only the piece to be produced and its corresponding data. It is also possible to display the defined work piece, or the entire machine work space.

Adjusting the display area makes it easier to visualize the data and make any adjustments.

Rotate data by 90 deg around Z

The NC data is rotated by 90°. This can quickly make sections of the data more easily visible.

Display - Graphic

By pushing the graphic button you enter the graphic mode
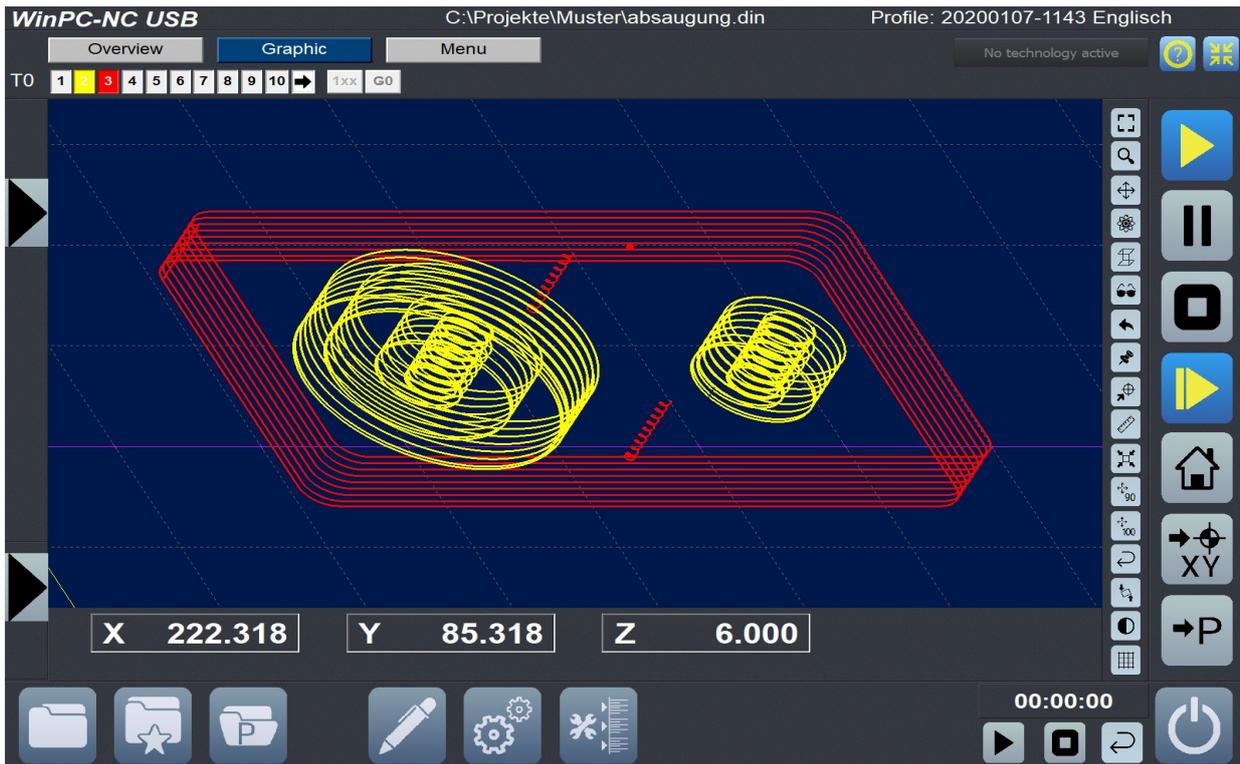
Graphic mode display

The axis coordinates are shown under the graphic window

The tool bar buttons (left to right):

Activate/Deactivate tools

When the NC data is initially displayed, each tool involved in the job is displayed with a different color. Each tool can be activated or deactivated from the display by clicking on its number. The *Display Area* will immediate be updated with the current selections.

In the example image, tools 2and 3 are activated.

If 30 tools are used you can select the next 10 tools with the arrow button

The jog window works the same as in the Overview mode.

Additional graphic functions in the graphic mode:

Set current machine position

The current machine position is represented by a small red dot in the *Display Area*. This function moves the current position to anywhere along a tool path. Simply click on the tool path and the current position is set to that location. The file zero point will be re-calculated.

Using this method, it is quite simple to position the working piece and the NC file with great accuracy.

Move machine to position

This function moves the machine to the position indicated by the cursor. Movement will be performed using a machine rapid move command. Care must be taken when using this command so as to not cause the machine to collide with anything.

Measuring function

Measuring sectors can easily be defined and limited by cursor and thus individual sections or distances can be directly measured in the diagram.
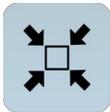
Center data in the working area

When working with engravings and millings it is sometimes advisable to center all of data to a pre-set area or on an *empty part*. The centering function accomplishes this. The zero point parameters will be re-calculated and no size change will occur.

Centering area means pre-defined work piece area. Any definitions and settings can be specified in parameter-coordinates or by manual move to the left bottom or top right corners as well as by the function keys [F5] and [F6].

Center data and Scale Data

Center data and Scale the work piece data by 90% or 100% in the working area .
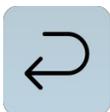
Undo scaling and centering

Previous scaling and centering operations are reverted. The previous parameters are restored.
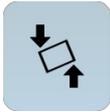
Angle measurement function / contacting 2 points

This button activates the function for measuring and start-up. Thus two clearly defined points in the diagram are approached by the machine and subsequently the respective position can be marked by the cursor in the diagram and twisted work pieces or templates can be correctly adjusted.

Toggle background color

Toggles the background color of the *Display Area* between light and dark. This is often useful when viewing difficult to see data.
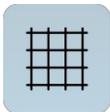
Grid on/off

Activates or deactivates the grid lines which serve as a useful reference. They are automatically scaled according to the drawing size. The grid size is indicated by the GX and GY data in the *Status Bar.*

Display - Menu

By pushing the menu button you enter the menu mode
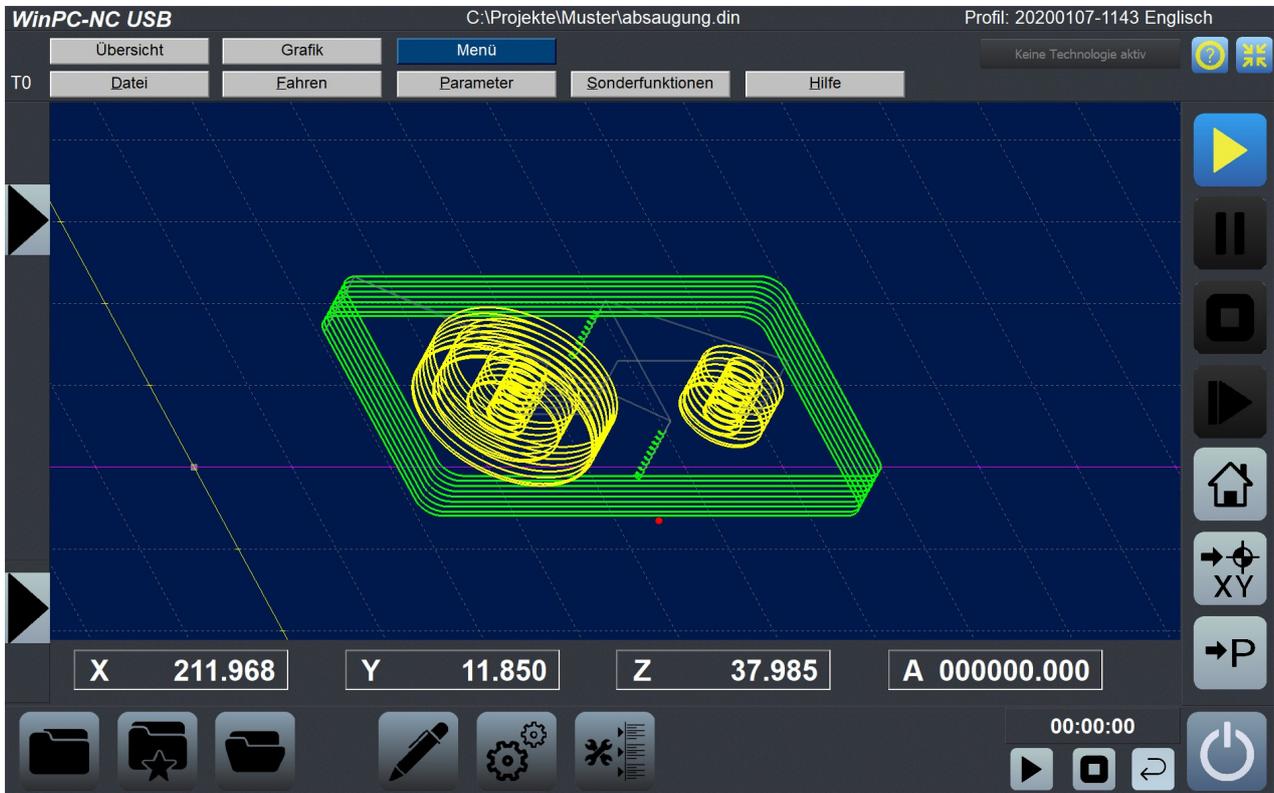
Touch-Ansicht mit geöffnetem Menü

Menu bar

<u>F</u>ile	<u>M</u>ove	<u>P</u>arameters	Special functions	<u>H</u>elp
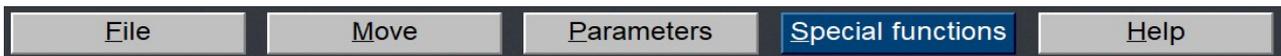
The menus and functions are the same as in the „Standard"
WinPC-NC

Pushing a menu button opens a dropdownlist with the possible functions.

WinPC-NC USB   C:\Projekte\Muster\absaugung.din   Profile: 20200107-1143 Englisch*

Overview	Graphic	Menu		Laser

T0 | File | Move | Parameters | Special functions | Help |

Signal test

Motor test

2D - CAM

Laser Cut Test

Laser Grayscale Test

Controller information

Rehoming check

Factory settings

Digitizing

Teach in

Spindle warm-up

Service code

| X | 222.318 | Y | 85.318 | Z | 6.000 |

00:00:00
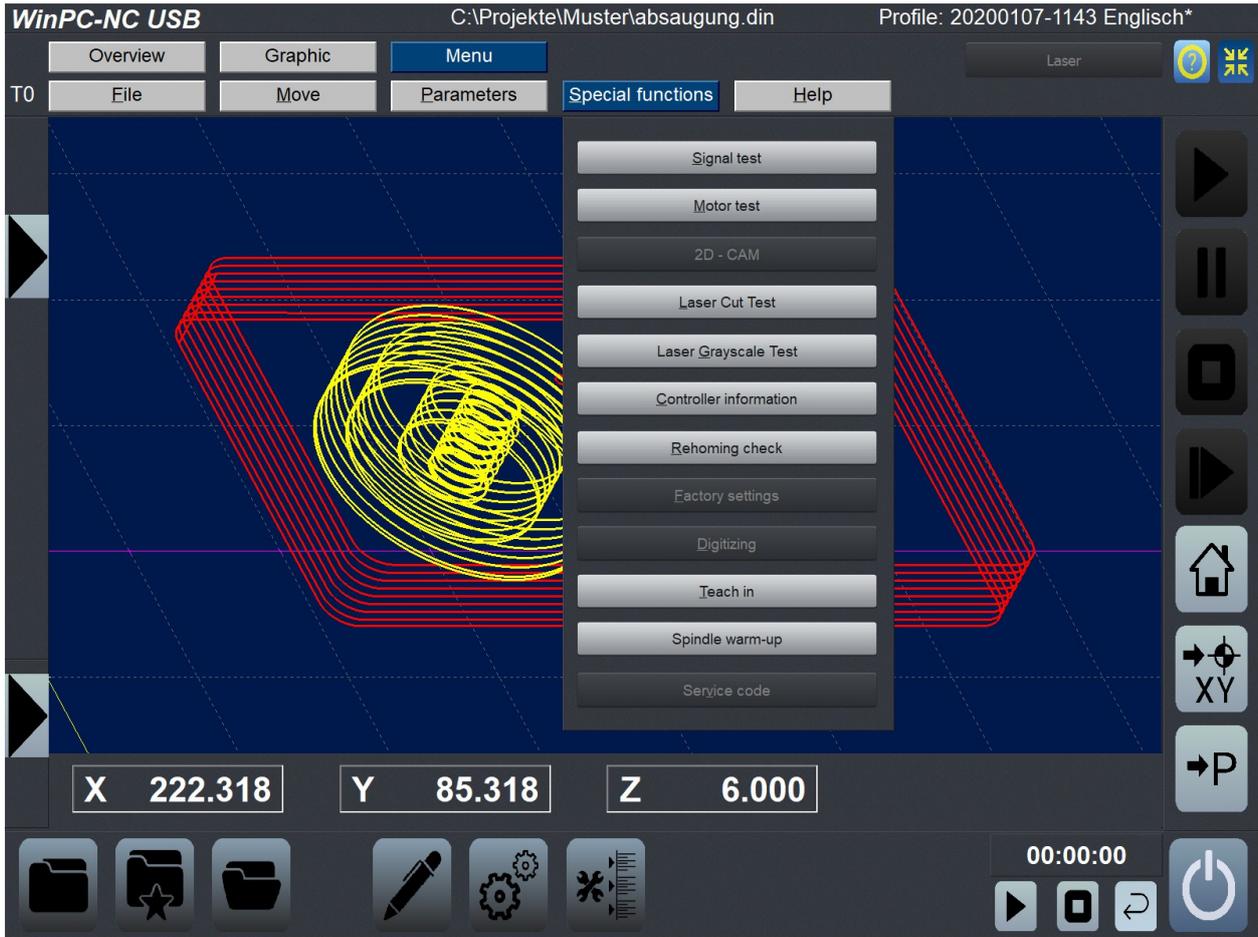
Touch menu mode with opened Special functions menu

2.4. Initial setup and test run

Executing an initial test is simple. Connect the NC machine motor signal wires according to the following assignments:
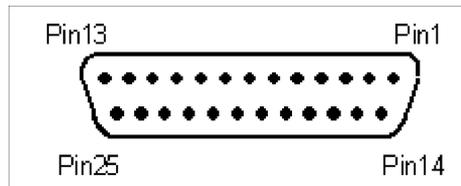
LPT connector at *nc100* or *ncUSB* or *CNCCON*, pinning

WinPC-NC uses the following signals for motor control:

Pin 2	**X axis motor direction**	**DIR X**
Pin 3	**X axis motor clock**	**CLK X**
Pin 4	**Y axis motor direction**	**DIR Y**
Pin 5	**Y axis motor clock**	**CLK Y**
Pin 6	**Z axis motor direction**	**DIR Z**
Pin 7	**Z axis motor clock**	**CLK Z**
Pin 8	**4th axis motor direction**	**DIR 4**
Pin 9	**4th axis motor clock**	**CLK 4**

Check and double check the wiring before using the MOTOR TEST or JOG function of **WinPC-NC**. Watch the machine carefully and be ready to use the emergency stop should an unexpected motion occur.

Jog dialog

Detailed instructions concerning the start-up procedure are given in a separate chapter.

2.5. Exiting *WinPC-NC*

You can exit *WinPC-NC* at any time by clicking on the program end button or by selecting FILE-EXIT.

Exiting of *WinPC-NC* with the exit Button.

3. Operating *WinPC-NC*

3.1. Graphical display of NC data

graphical dis-
play of the
NC files

WinPC-NC fills the *Display Area* with a graphical representation of a NC file as soon as its data is loaded. All contours or vectors are displayed in color.
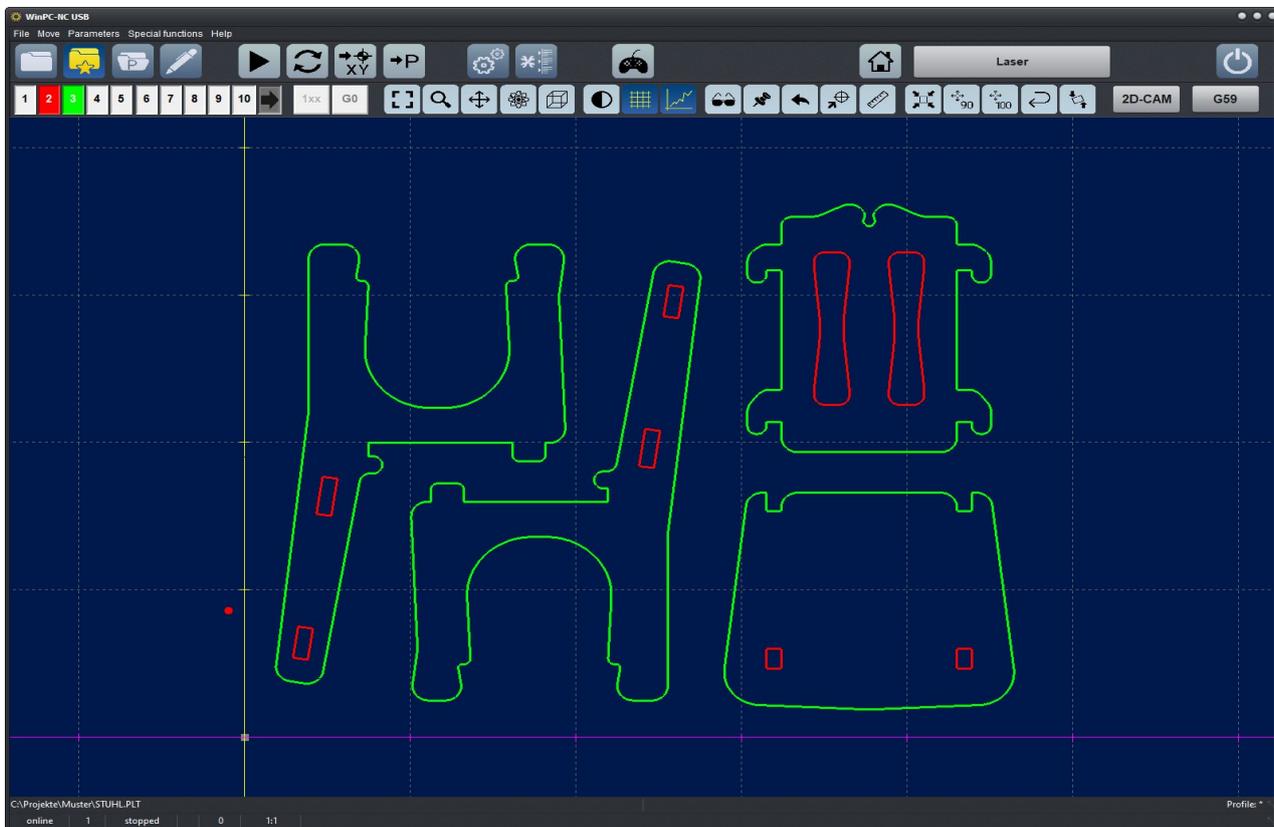
Loaded NC file with standard-view

Graphic dimensions can easily be sized up by displayed grid patterns and its distances are indicated below the diagram.

The workpiece zero point is marked as a small grey quadrat and the current position of the machine as a small red dot. However, this position is only visible if *WinPC-NC* knows the current position or referencing has been made.
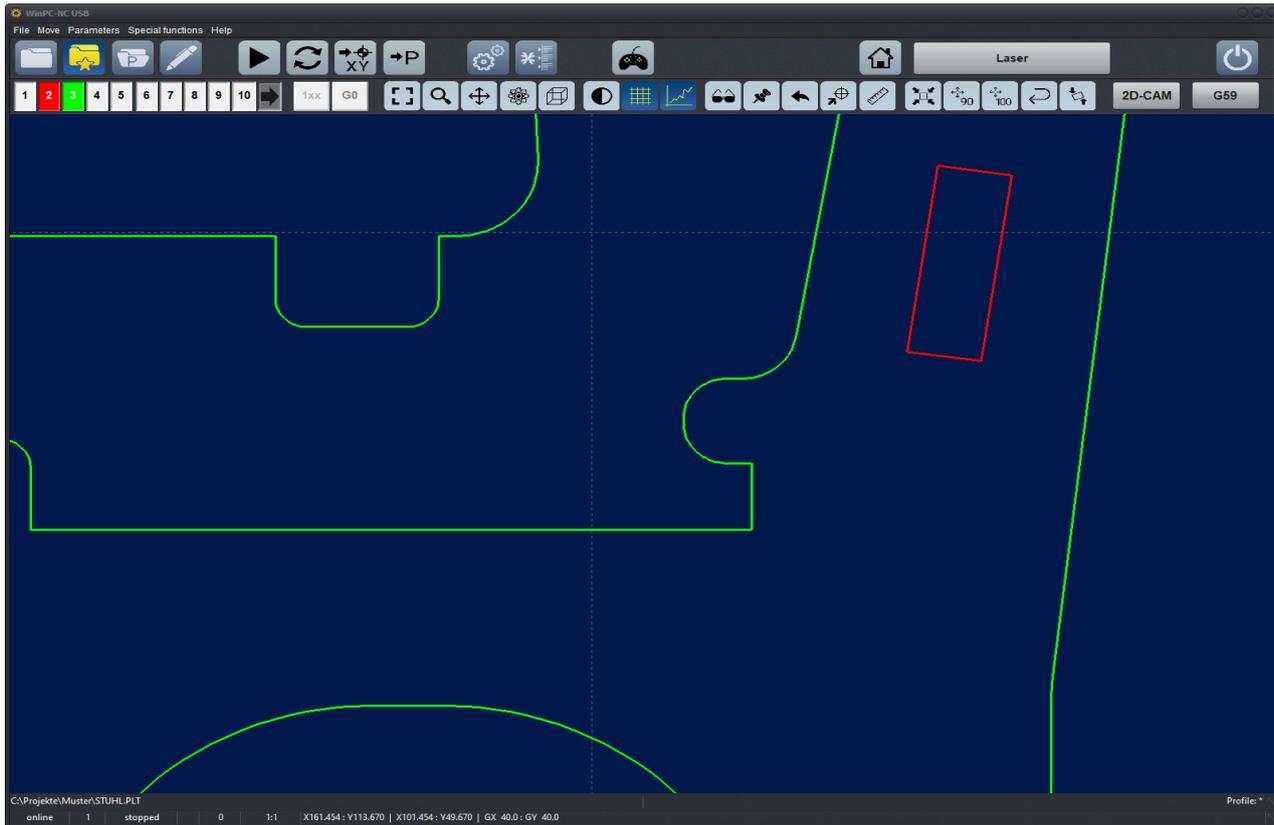
Zoomed and moved view

The initial view presents the NC data looking down at the XY pla-
ne. Other perspectives may be selected; the NC data can be rota-
ted, scrolled, and zoomed. Measurements can be estimated by
referring to the grid-lines

The tool bar buttons (left to right):

Zoom the data

The NC data may be zoomed - *magnified* - by activating this
mode. The mouse cursor will change to notify you that zooming
is in effect. Click the left mouse button in the *Display Area* and
moving the mouse around will cause the NC data to zoom.

Zooming can also be done by moving the mouse scroll.

Scroll the data

The NC data may be scrolled – *moved around* - by activating
this mode. The mouse cursor will change to notify you that scroll-
ing is in effect. Click the left mouse button in the *Display Area*
and moving the mouse around will cause the NC data to scroll.

Scrolling can also be done by moving the mouse around while in
the *Display Area* and keeping the right mouse button pressed.

Reset display view

Restores the Display Area to its initial condition. Any changes to the NC data will be lost. If a mode is selected, it will be deselected.

Set current machine position

The current machine position is represented by a small red dot in the *Display Area*. This function moves the current position to anywhere along a tool path. Simply click on the tool path and the current position is set to that location. The file zero point will be re-calculated.

Using this method, it is quite simple to position the working piece and the NC file with great accuracy.

Move machine to position

This function moves the machine to the position indicated by the cursor. Movement will be performed using a machine rapid move command. Care must be taken when using this command so as to not cause the machine to collide with anything.

Rotate data by 90 deg around Z

The NC data is rotated by 90°. This can quickly make sections of the data more easily visible.

Select the display area

Display only the piece to be produced and its corresponding data. It is also possible to display the defined work piece, or the entire machine work space.

Adjusting the display area makes it easier to visualize the data and make any adjustments.
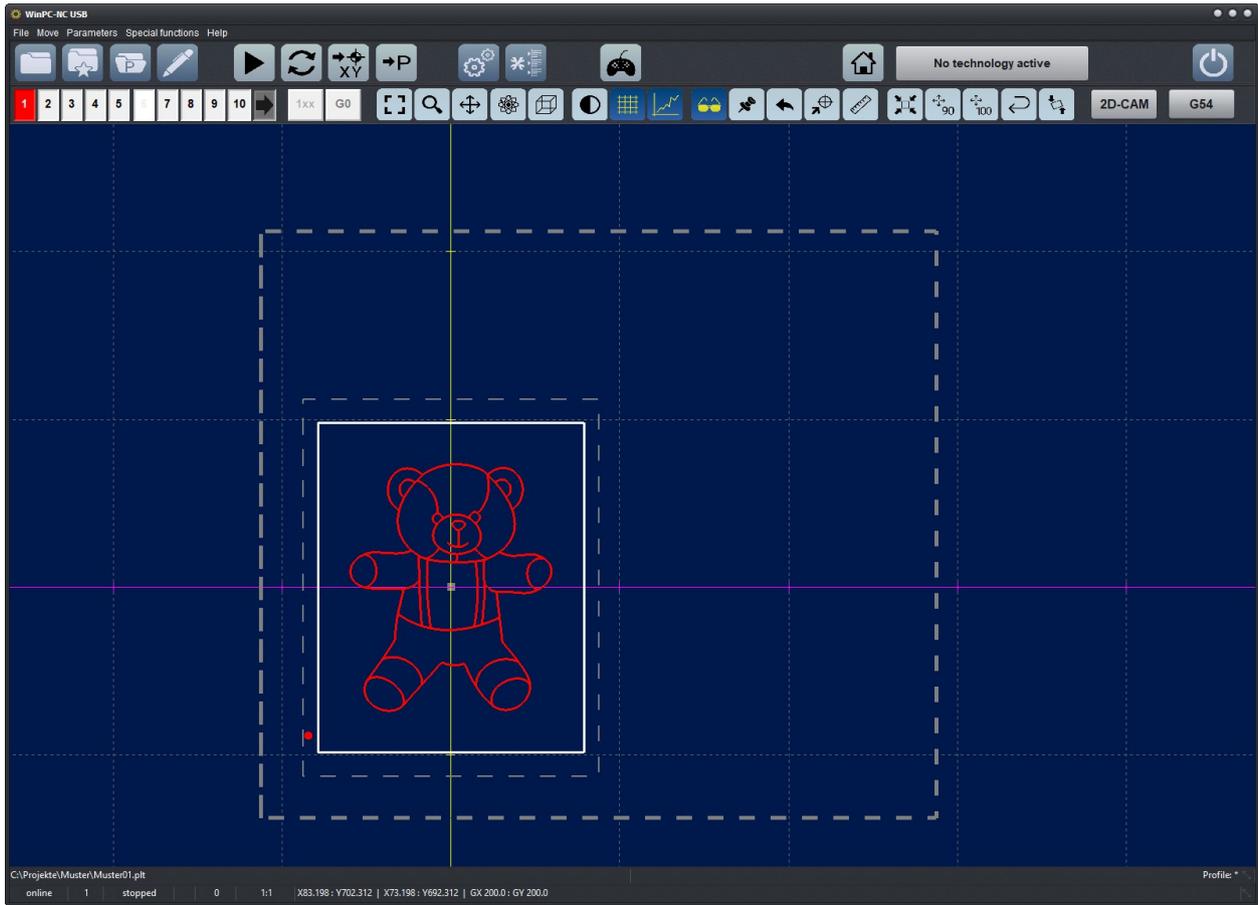
Graphical view with machine table and defined workpiece area
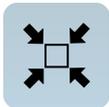
Center data in the working area

When working with engravings and millings it is sometimes advisable to center all data to a pre-set area or on an *empty part*. The centering function accomplishes this. The zero point parameters will be re-calculated and no size change will occur.

Centering area means pre-defined work piece area. Any definitions and settings can be specified in parameter-coordinates or by a manual move to the left bottom or top right corners as well as by the function keys F5 and F6 .
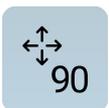
Center and scale Data

Center and scale the work piece data by 90% or 100%.
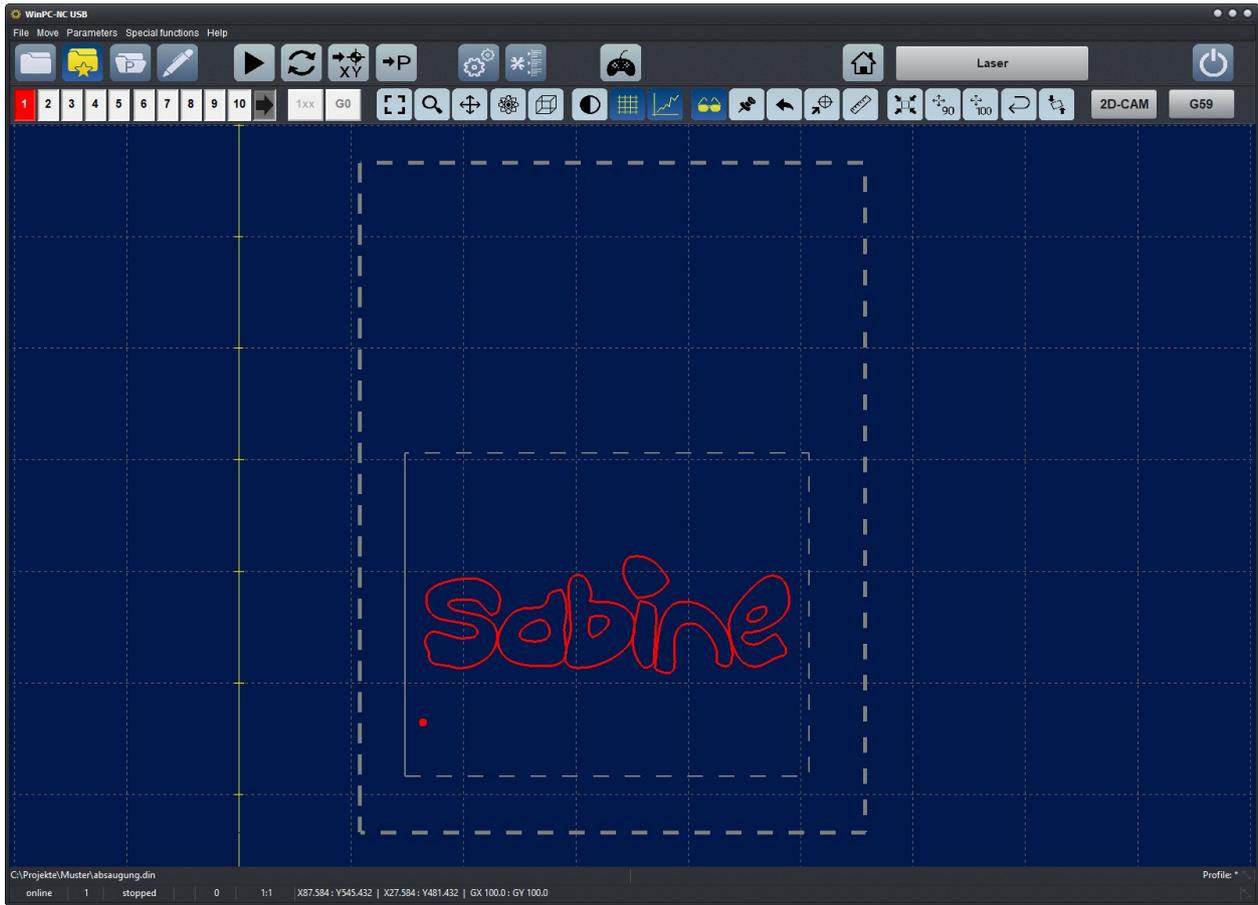
Data centered and scaled to 90%

Undo scaling and centering

Previous scaling and centering operations are reverted. The previous parameters are restored.

Toggle background color

Toggles the background color of the *Display Area* between light and dark. This is often useful when viewing difficult to see data.
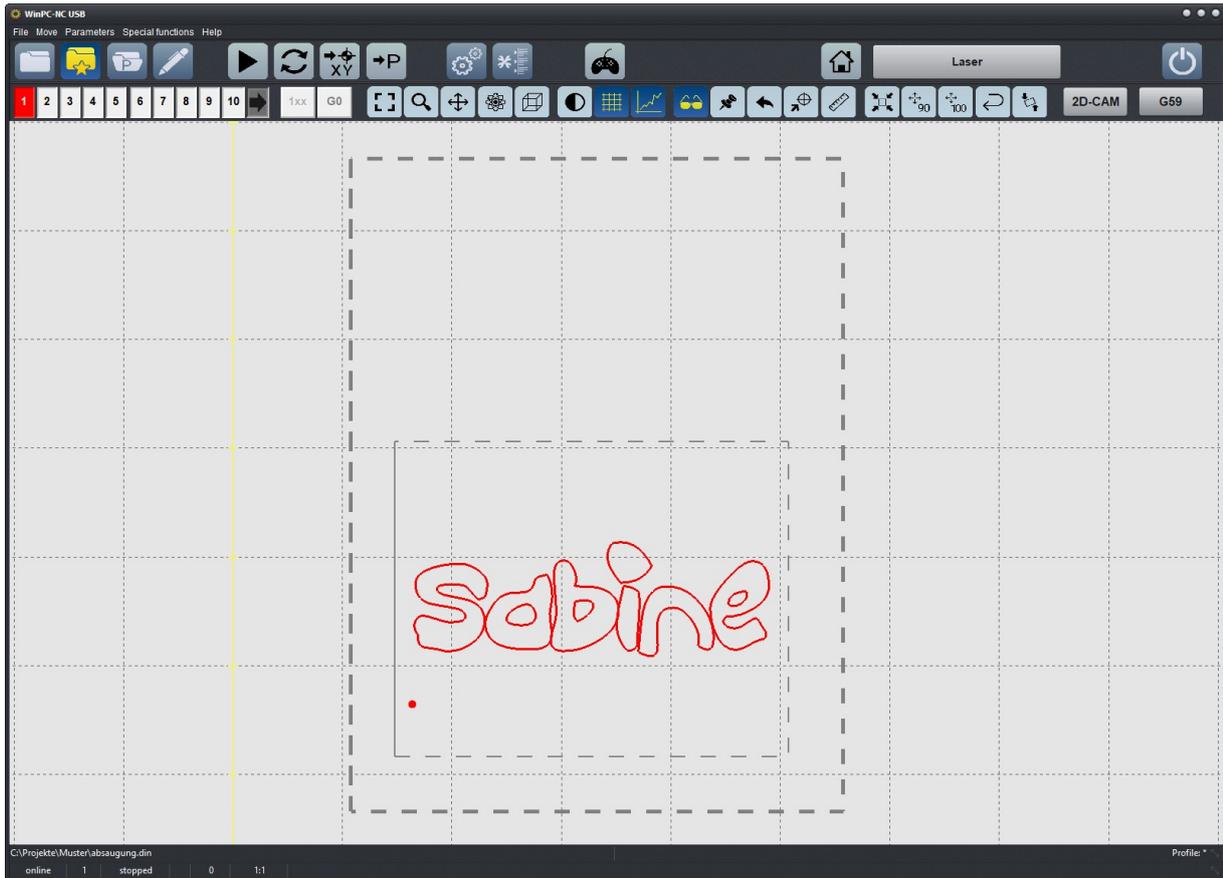
Backgroundcolor bright
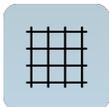
Grid on/off

Activates or deactivates the grid lines which serve as a useful reference. They are automatically scaled according to the drawing size. The grid size is indicated by the GX and GY data in the *Status Bar.*

Cycle axis display

Cycles the display of the axes. The axes will be hidden, only XY will be displayed, or XYZ will be displayed.

Standard perspective views

Clicking this button cycles the display through 4 useful perspectives.

WinPC-NC
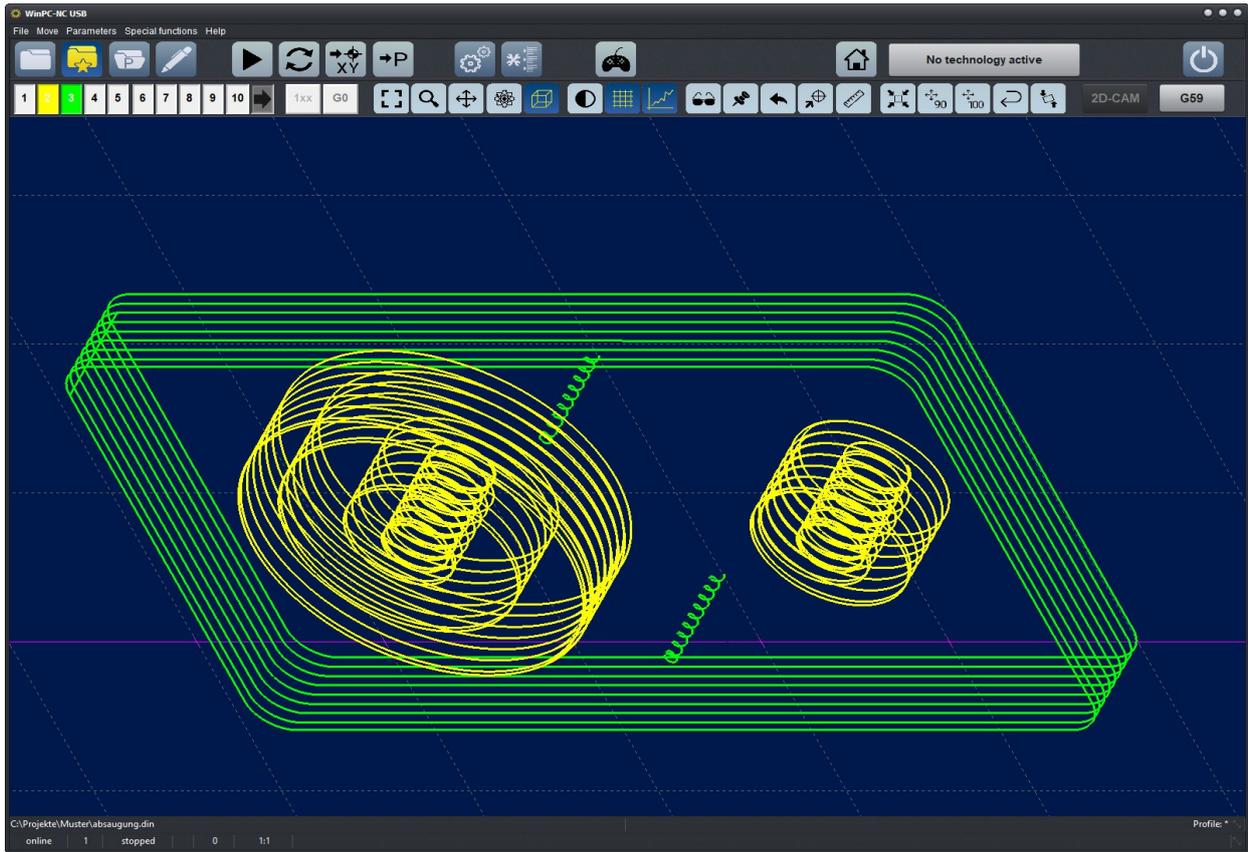
Perspective view
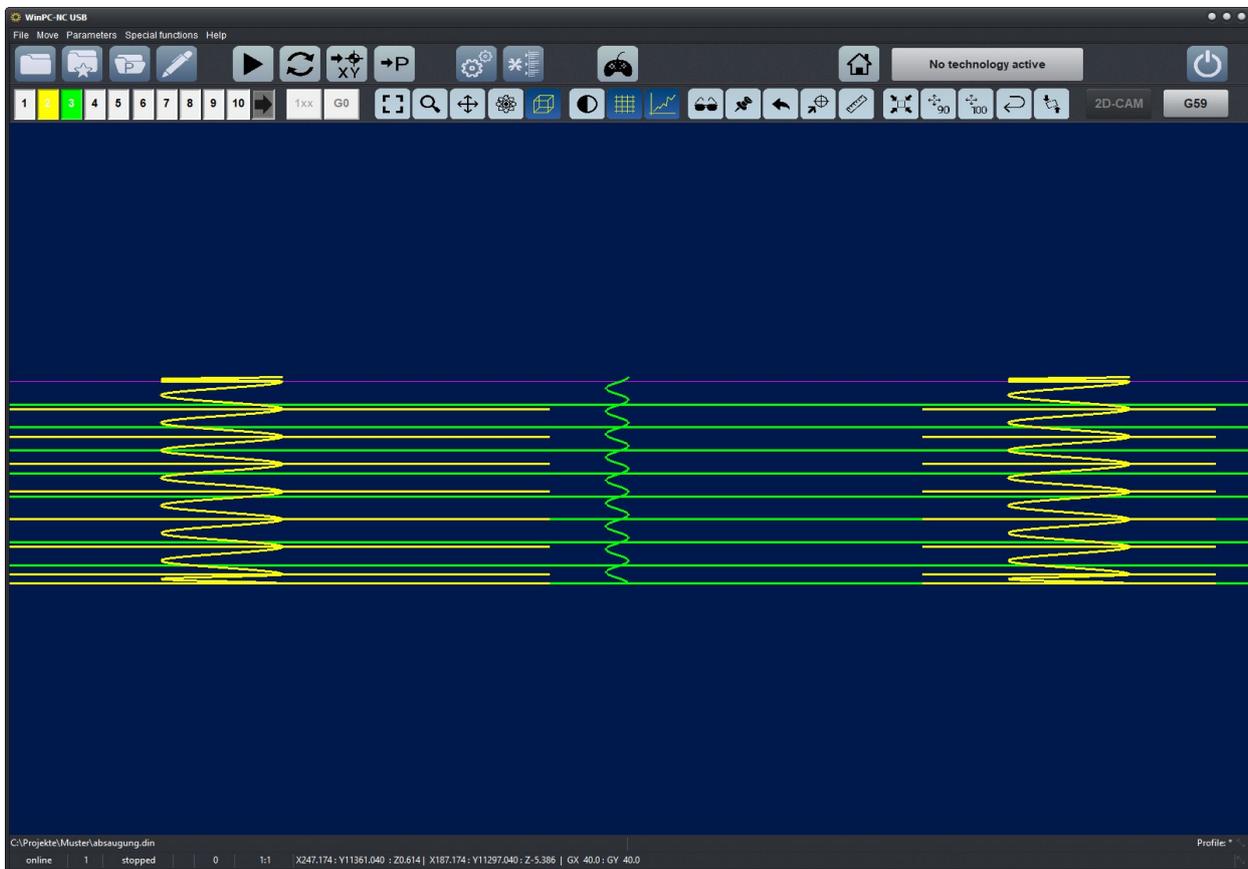
View from the left to the Y- and Z- level with precise display of the mouse position
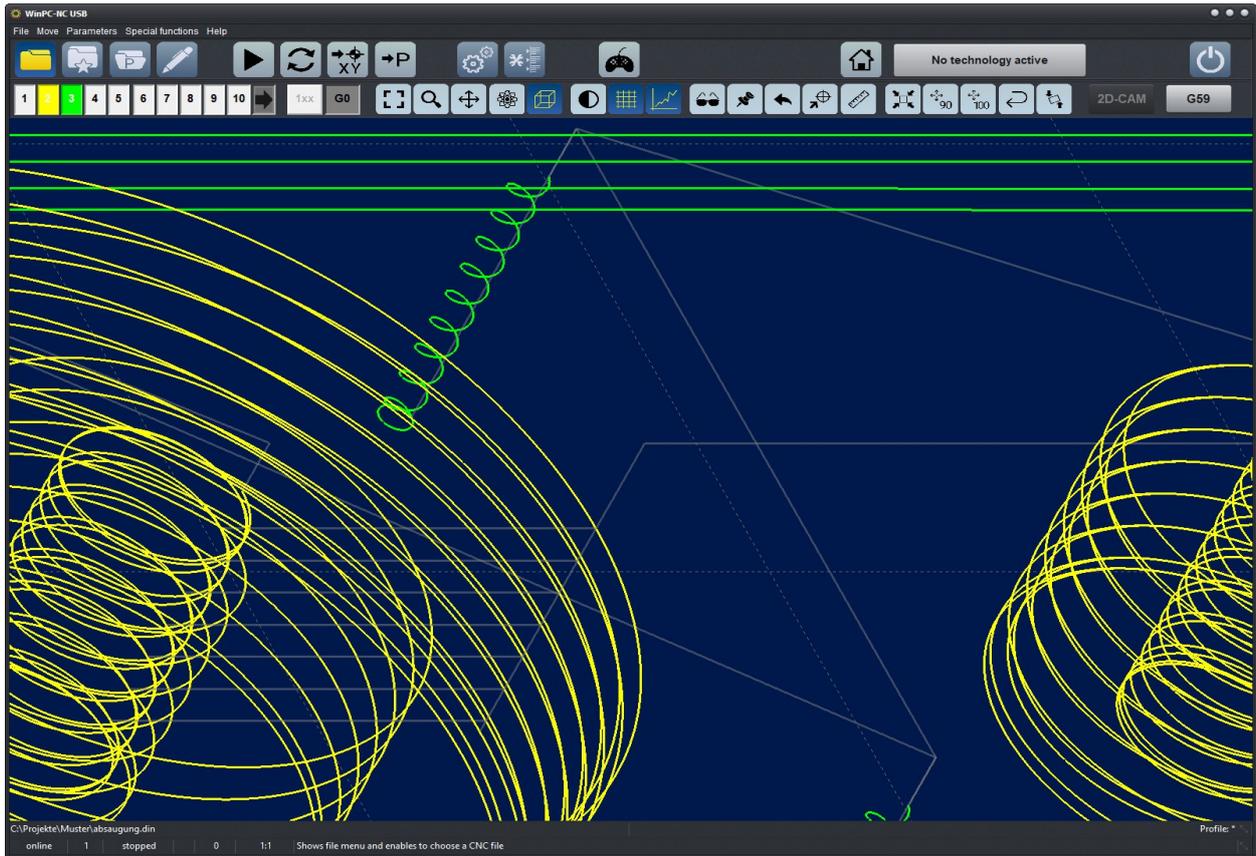
Orbit-function - easily rotating the data

Orbit the data

The NC data may be orbited - *rotated around* - by activating this mode. The mouse cursor will change to notify you that orbiting is in effect. Click the left mouse click in the *Display Area* and moving the mouse around will cause the NC data to be orbited.

Toggle empty moves

Unproductive empty movements will be displayed or suppressed.

1xx **Display or hide original distances**

Subsequently to a radius compensation with the internal 2D-CAM function original vectors and lines can be displayed for control or hidden.
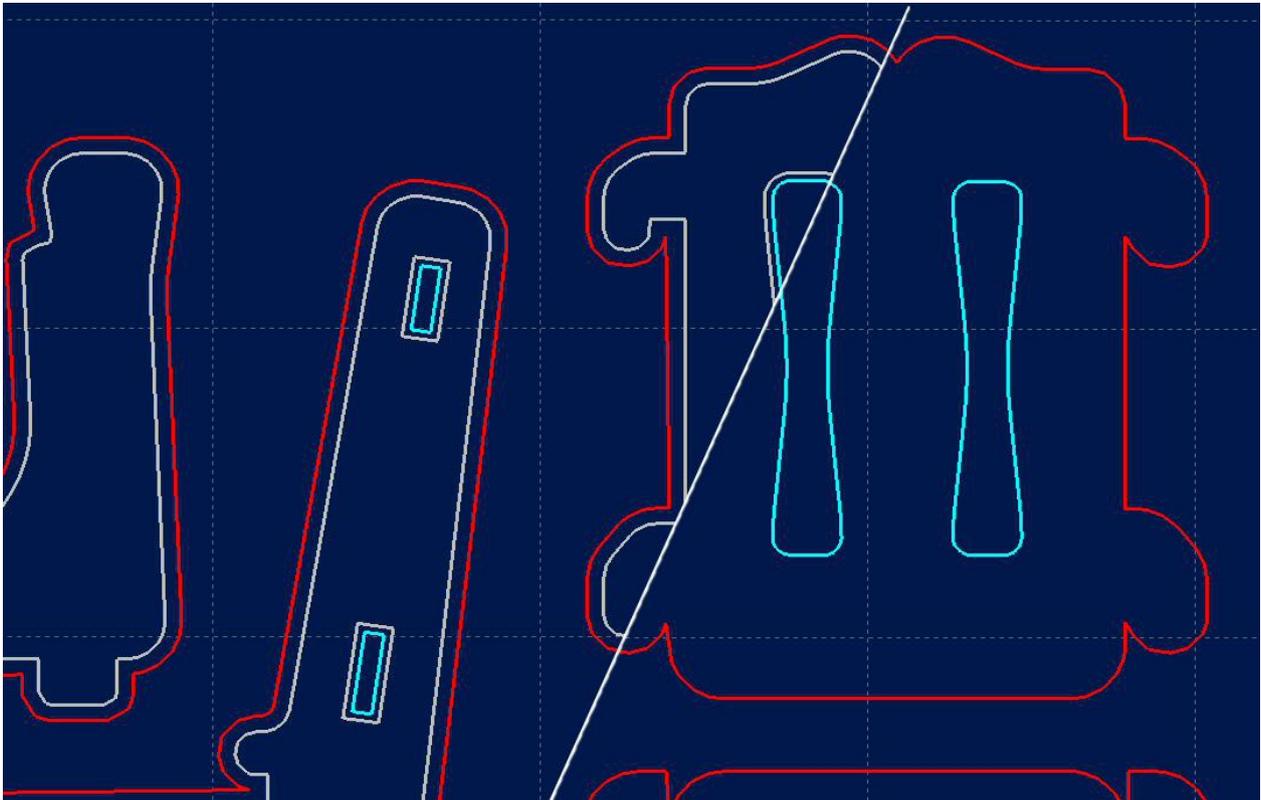
Display with and without original distances

Activate/Deactivate tools

When the NC data is initially displayed, each tool involved in the job is displayed with a different color. Each tool can be activated or deactivated from the display by clicking on its number. The *Display Area* will immediate be updated with the current selections.

In the example image on the left, tools 1, 2 and 4 are activated. Tool number 3 is deactivated and tool number 5 is not involved in this job.

2D CAM functions

2D-CAM

WinPC-NC can apply CAM functions to many 2D data formats (e.g. HPGL, EPS/AI, DXF(2D), and drilling). The 2D data can be cleaned up, sorted, and optimized. Tool diameter compensation can be specified.

A detailed step by step guide can be found in the next chapter.
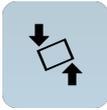
Angle measurement function / contacting 2 points

This button activates the function for measuring and start-up. Thus two clearly defined points in the diagram are approached by the machine and subsequently the respective position can be marked by the cursor in the diagram and twisted work pieces or templates can be correctly adjusted.

Measuring function

As it is easy to define measuring sections via cursor, individual sections or distances can be directly measured in the diagram.
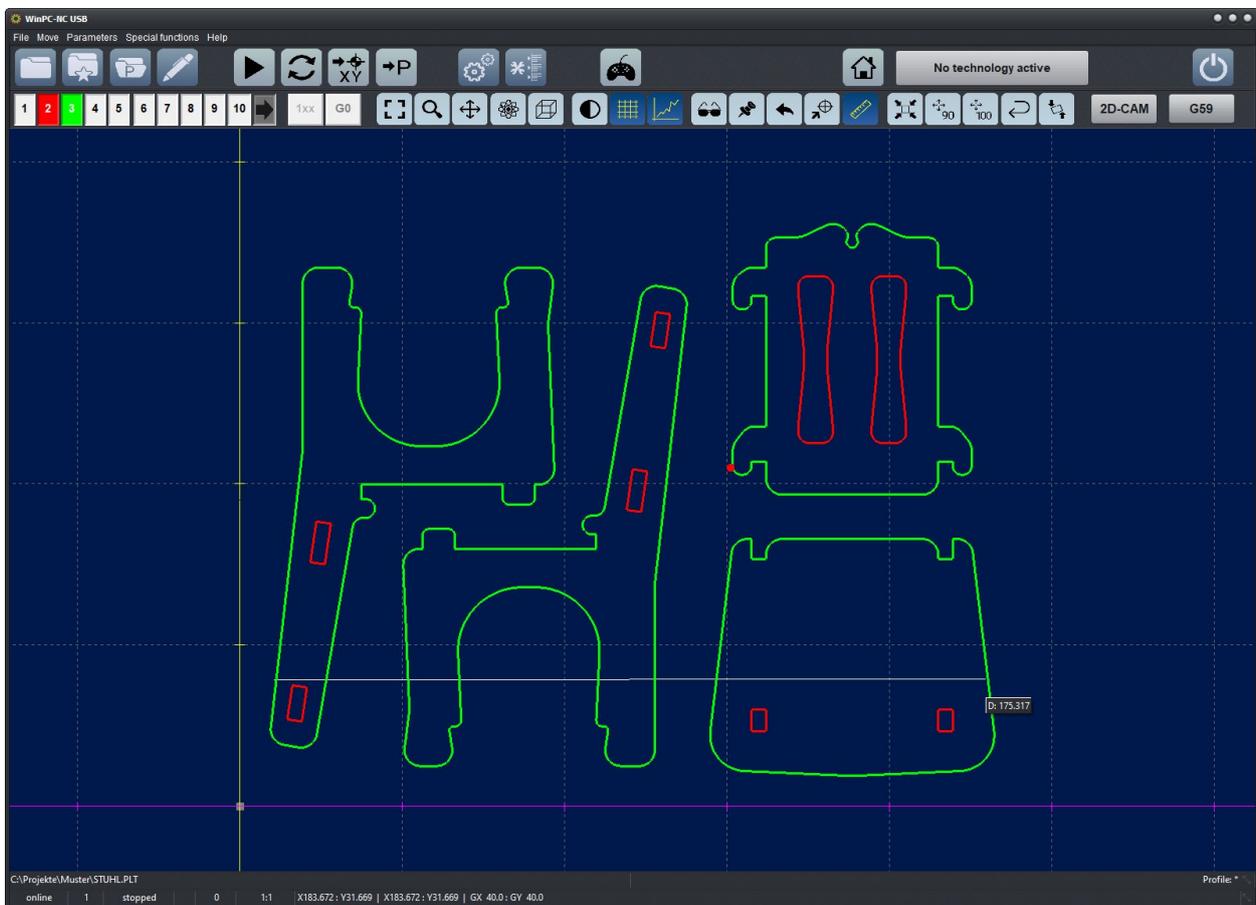
Measuring function with display of defined measuring line by cursor

WinPC-NC allows many of its windows to be *torn* from the initial place and positioned anywhere you like.

The position of any *torn* windows is retained from session to session.
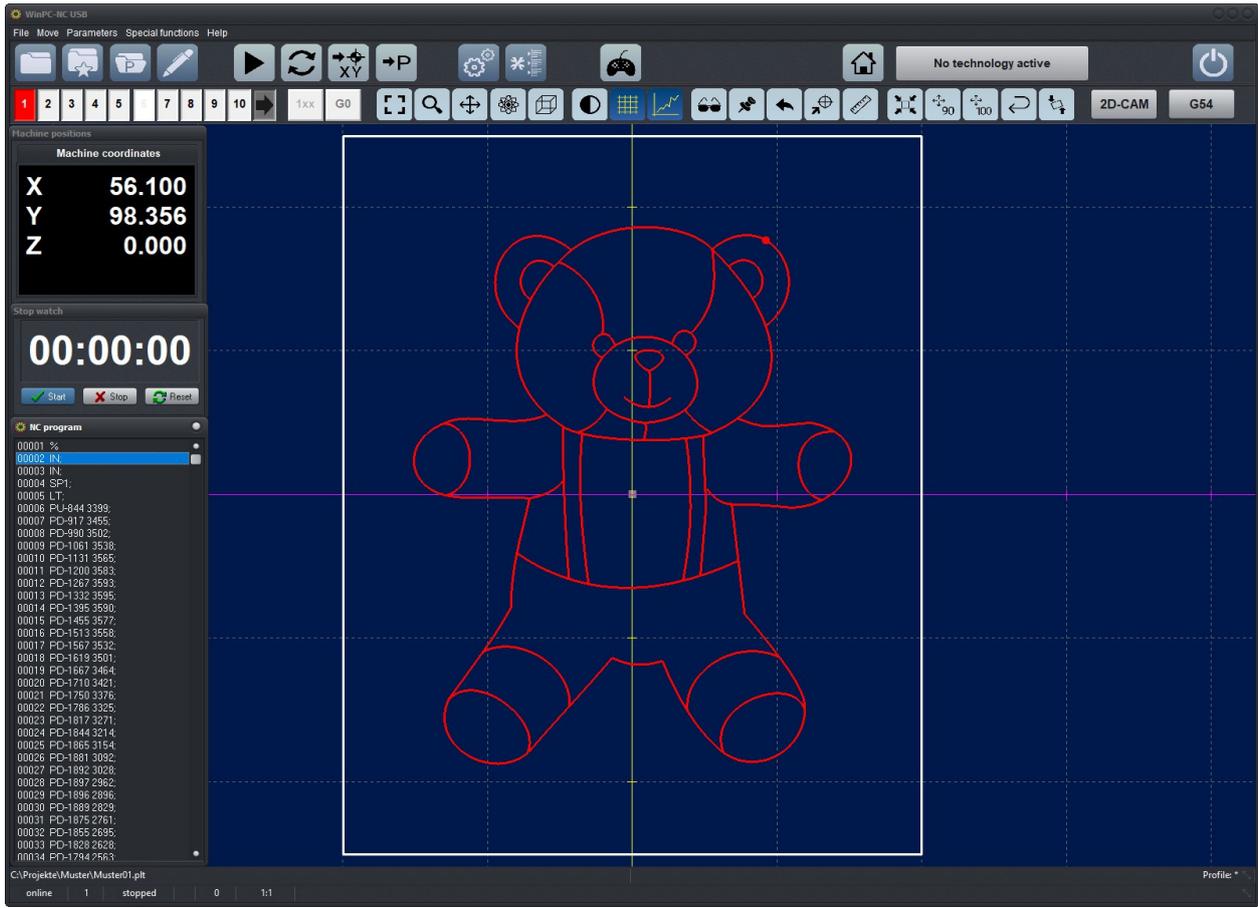
Different windows arranged side by side

3.2. Drop-down menus and function keys

easy to use interface

WinPC-NC has an easy to use user interface. All functions may be found in drop-down menus. Many of the menu entries have an accelerator, a function key assigned to make rapid access possible.

The drop-down menus are functionally grouped - file, movement, parameters, special functions, and help.

Opening or activating the menu system is made by clicking the menu item or one of the hotkey buttons of the individual menu.

additional function keys

Important functions can also be activated using function keys. Function key assignment is fixed and indicated in the menu functions.

The most important function keys are:

`F1`	Activate the help system
`F2`	Load NC file
`shift-F2`	Load parameter data
`F3`	Start job from beginning
`shift-F3`	Start job from defined point
`F4`	Move to XY origin
`F5`	Jog (mouse or pendant)
`shift-F5`	Jog (joystick)
`F7`	Load file into editor
`F8`	Start machine initialization
`F9`	Move to parking position
`F10`	Open pull down menu

3.3. The individual menus

Each menu and its items are explained in detail.

Some menu items may not be available at all times (e.g. joystick control when no joystick is configured).

3.3.1. FILE menu

The FILE menu is used for loading files for analysis and processing, editing files, and to exit *WinPC-NC*.

```
┌─────────────────────────────────────┐
│                                     │
│  Open...                    F2      │
│  Open without parameters...         │
│  Last opened...                     │
│  Editor                     F7      │
│  Show installation directory        │
│                                     │
│  Exit                               │
│                                     │
└─────────────────────────────────────┘
```

Press the ⌊alt-D⌋ shortcut to open the file menu.

FILE - OPEN

File selected by menu   This menu item loads the selected NC file and any previous parameter data.

In the selection screen area you can change drives and directories and activate filters for specific file extensions. File selection can also be made by function key ⌊F2⌋ or by clicking the *Open* button.
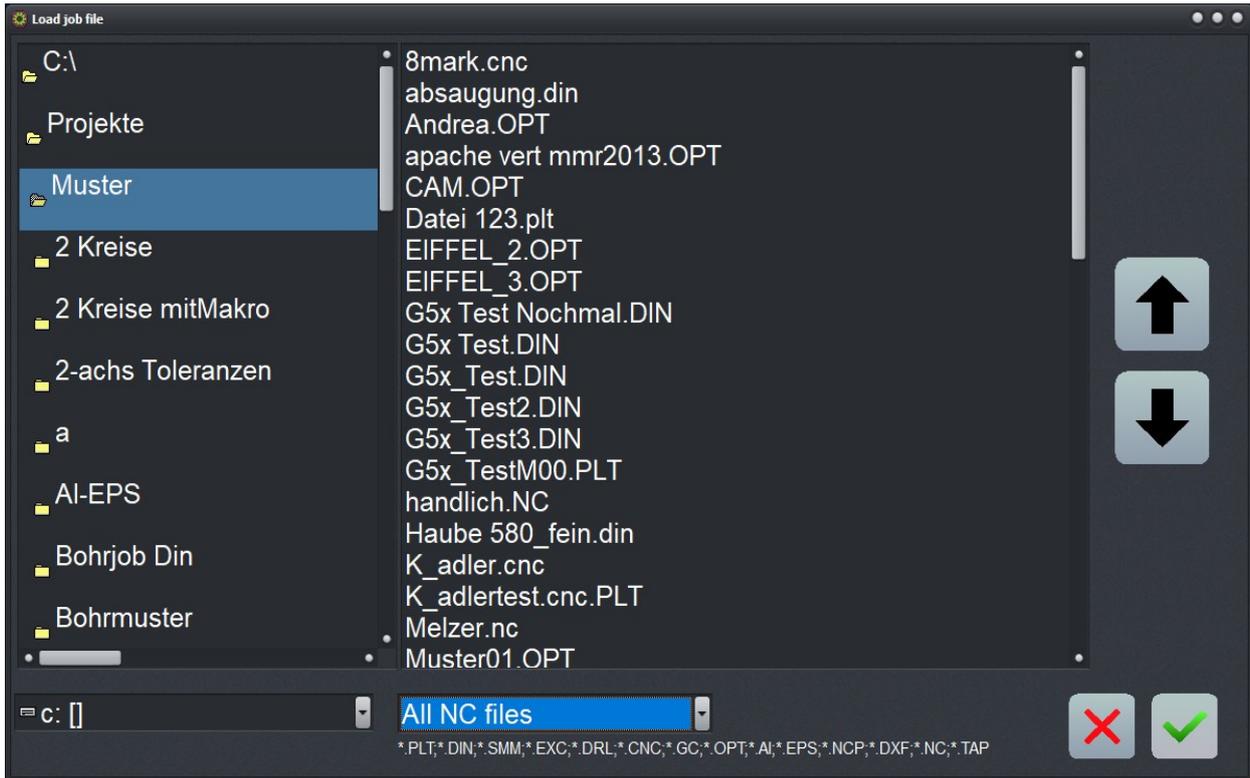
File selection menu with drive and path and file list

If you want to use the standard Windows dialog for opening or saving a file, you have to set the entry WINDIALOG=0 on WINDIALOG=1 in the WinPCNC.WPI file (in the starting directory of WinPC-NC).

Loading the NC file with previous used parameters

Preselection can be made by a filter. Either NC files with standard extensions are displayed or all files. By loading older files all parameters are activated which probably have been stored previously for this project.

Graphical preview of NC files

The graphic preview in *WinPC-NC* is active immediately upon selecting a NC file. All contours and drillings are visualized in the relevant tool color.

FILE - OPEN WITHOUT PARAMETERS

This item loads the selected NC file but ignores any previous parameter data. All active parameters remain valid.

FILE - LAST OPENED

This item will display the last NC file that was opened. Click on the file name to load it. Once the NC data is loaded the *Display Area* will present the data graphically.

FILE - EDITOR

Postprocessing of the NC-Files

This item activates the NC file editor. The **WinPC-NC** editor will be used when an external editor has not been specified.
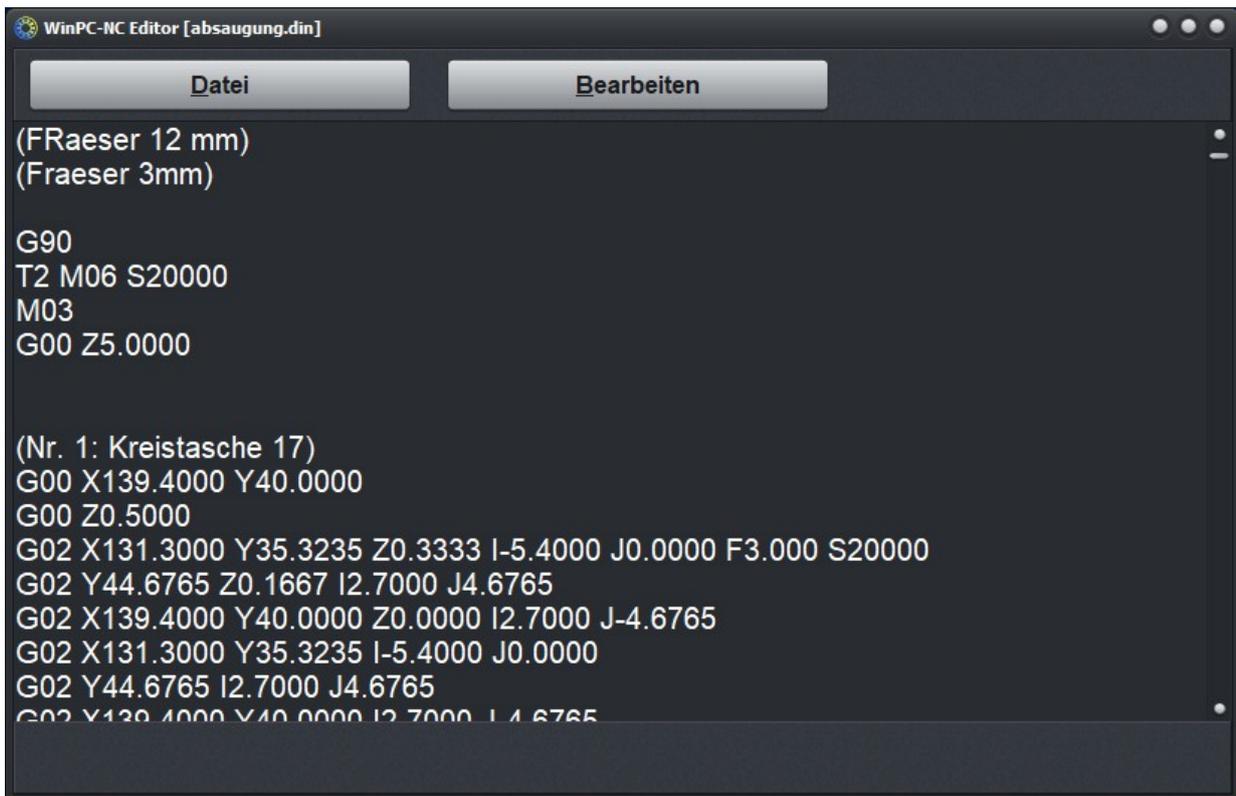
Opened editor with current file

The editor can also be activated by hitting [F7] or by clicking the editor button.

FILE - SHOW INSTALLATION DIRECTORY

This function opens the explorer and the installation directory displays the current version of **WINPC-NC** This simplifies the procedure of backing up or sending files, e. g. parameters or protocols

FILE - EXIT

This item exits *WinPC-NC.* Any NC machine activity will be terminated; all files will be closed. Clicking the exit button has the same effect.

3.3.2. MOVE menu

Functions for moving your machine

The MOVE menu contains those functions which are used for controlling the machine and manipulating tools.

Start	F3
Start from...	shift-F3
Start single steps	
Zero point XY	F4
Park position	F9
Jog	F5
Joystick jogging	shift-F5
Initialize machine	F8
Select tool	

Press the `alt-F` shortcut to open the file menu.

MOVE - START

This item will start the loaded job from the beginning.

Pressing `F3` or clicking the START button has the same effect.

Travel commands induce *WinPC-NC* to control the X and Y motors. Motor Z moves up or down if you activate the relevant command for moving the tool. Three or even four axes can move at the same time in 3D files.

Progress indicator

The current tool position, marked by a red dot in the *Display Area*, will be updated in real-time.

A Job running window will appear. The percentage of the job that has been completed is updated in real-time.

Clicking the STOP button or by pressing the `ESC` key will terminate the job. The machine will immediately stop moving (no steps will be lost), the spindle will be turned off, and any coolant distribution will be turned off.

The Job running window with speed and spindle override

Feedrate and spindle override

The feed and spindle rates may be adjusted, up or down, in steps of 5%. The minimum rate is 10%; the maximum 200%.
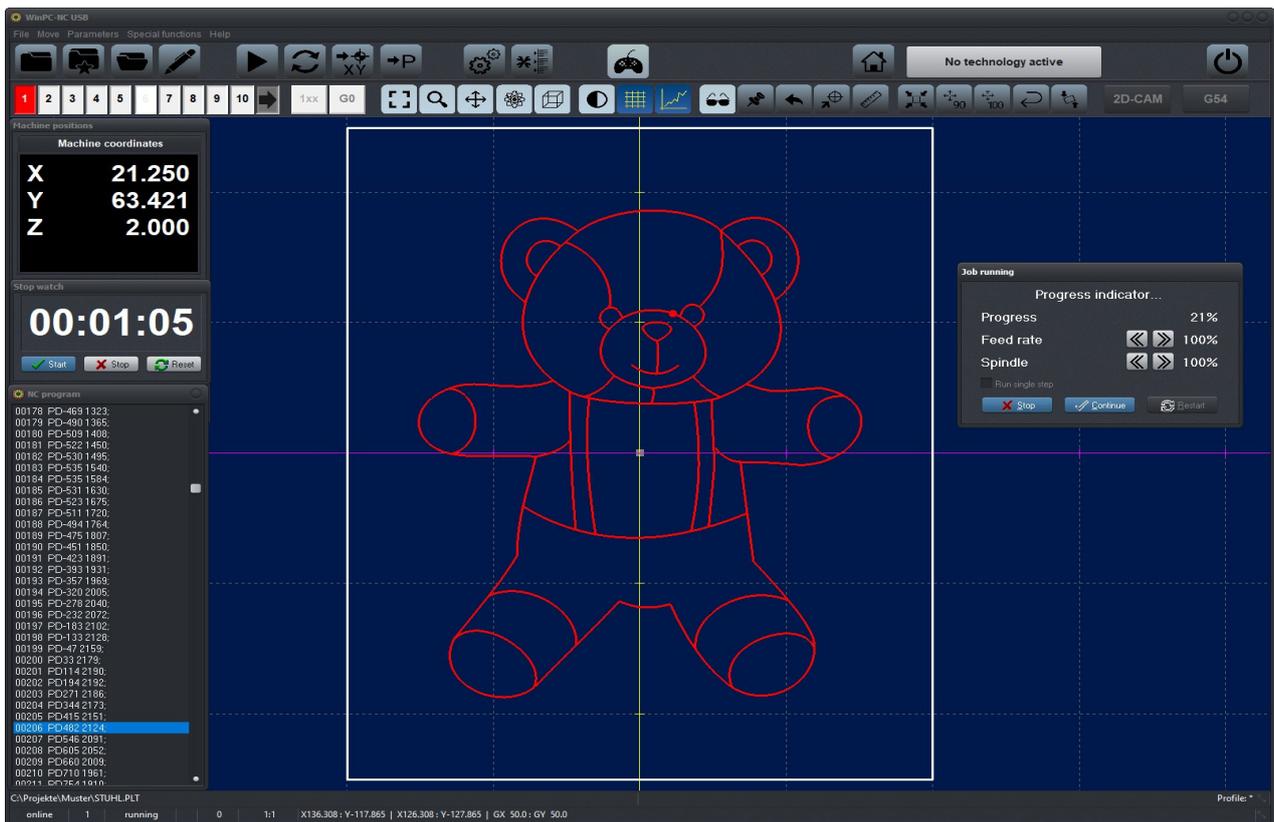
Started job (standard view) with additional windows (machine-coordinates, stop watch, NC window)

Display with side panel means nothing but changed data content and tolerates the same conditions.
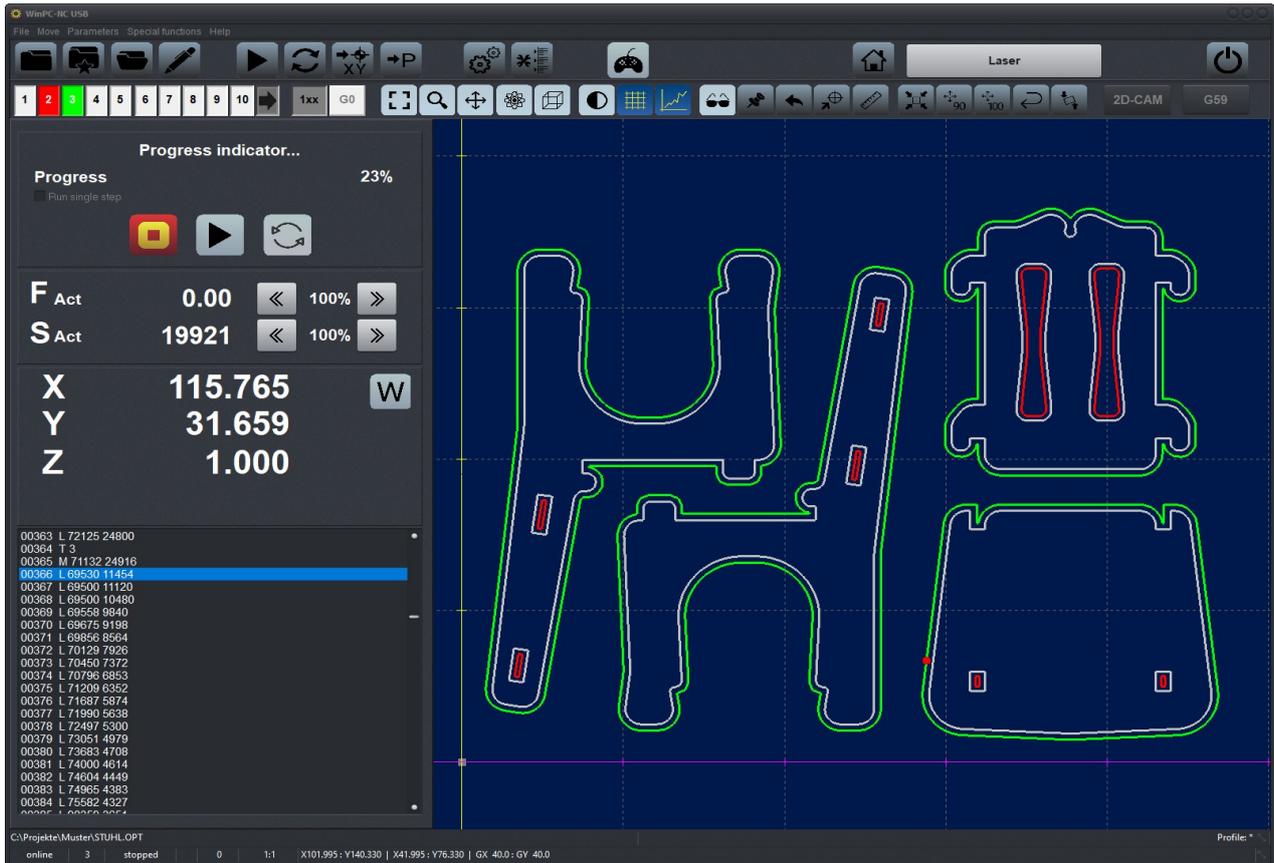
Running job with sidepanel

Clicking the STOP button or by pressing the (ESC) key will termi-
nate the job. The machine will immediately stop moving (no steps
will be lost), the spindle will be turned off, and any coolant distri-
bution will be turned off.

Continue inter-
rupted job

By rebooting the system after a breakdown *WinPC-NC* wants to
know whether the interrupted process should be continued from
the breakpoint or started again. During the stop parameters can
be changed, tools can be cleaned or changed and it is possible
to execute jog move and homing.

Continue a paused job

The ability to CONTINUE (from the Job running window) is controlled by a setting in the PARAMETERS menu.

The START-FROM menu item (see below) can be used to continue a job.

MOVE - START FROM...
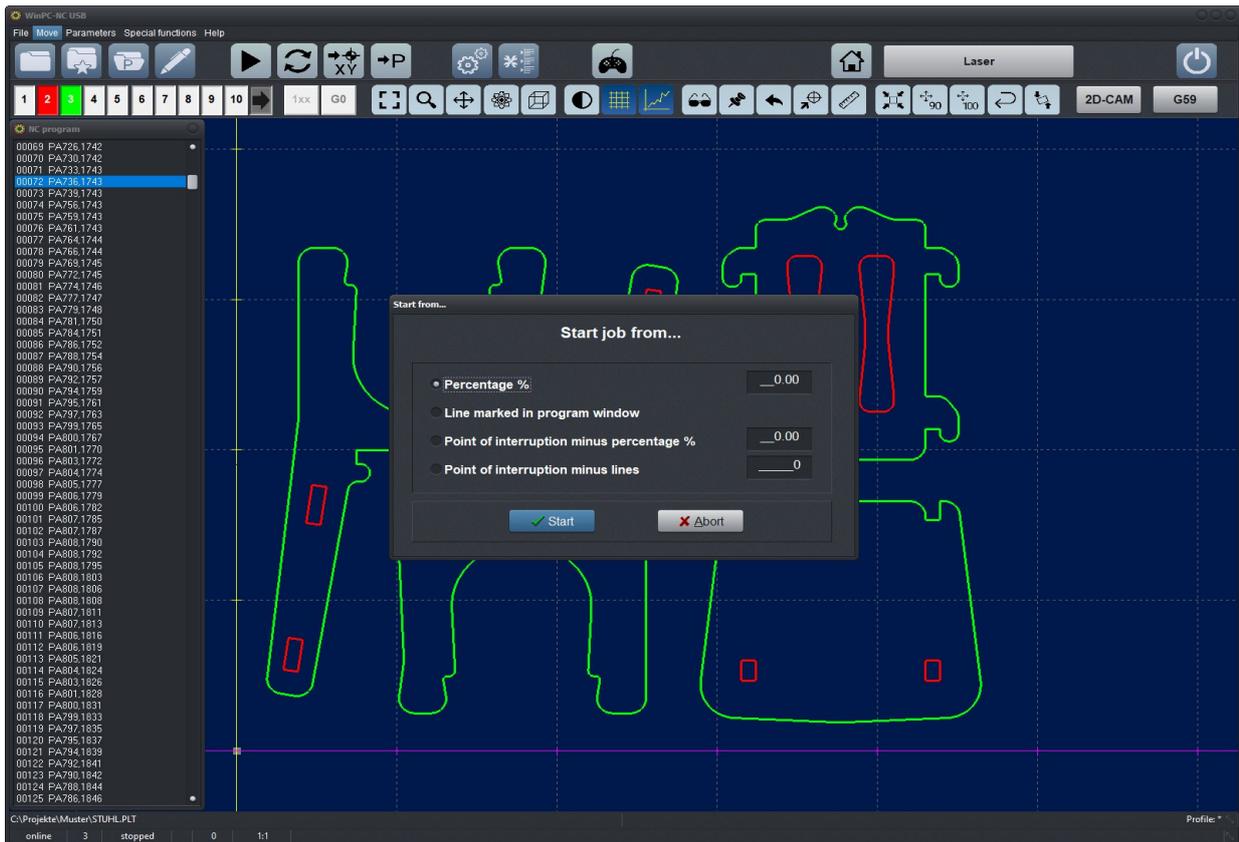
Job starts from the beginning or a designated place.

This item allows the loaded job to be started from the beginning or a designated place. There are four ways to choose the designed place.

- Start from the specified job percentage

- Start from the line marked in the program window

- Start from the point of interruption less a percentage

- Start from the point of interruption less a number of lines

WinPC-NC will determine the point to begin, move to it, and process the job from there. If the job is being continued, the spindle and coolant will also be restored before motion begins.

MOVE - START SINGLE STEP

Sometimes it is advisable to start the job slowly, i. e. without maximum speed and without full use of tools, but to approach the material by individual program steps in order to check the exact positions.

This item runs the loaded job one NC instruction at a time. One NC instruction will be performed and *WinPC-NC* will wait for the CONTINUE or CANCEL button to be hit.

Moving one NC instruction at a time is useful for debugging a problem or slowing the machine down so one can see exactly what is going on.

Uncheck the single step check box and hit CONTINUE to resume full speed execution.
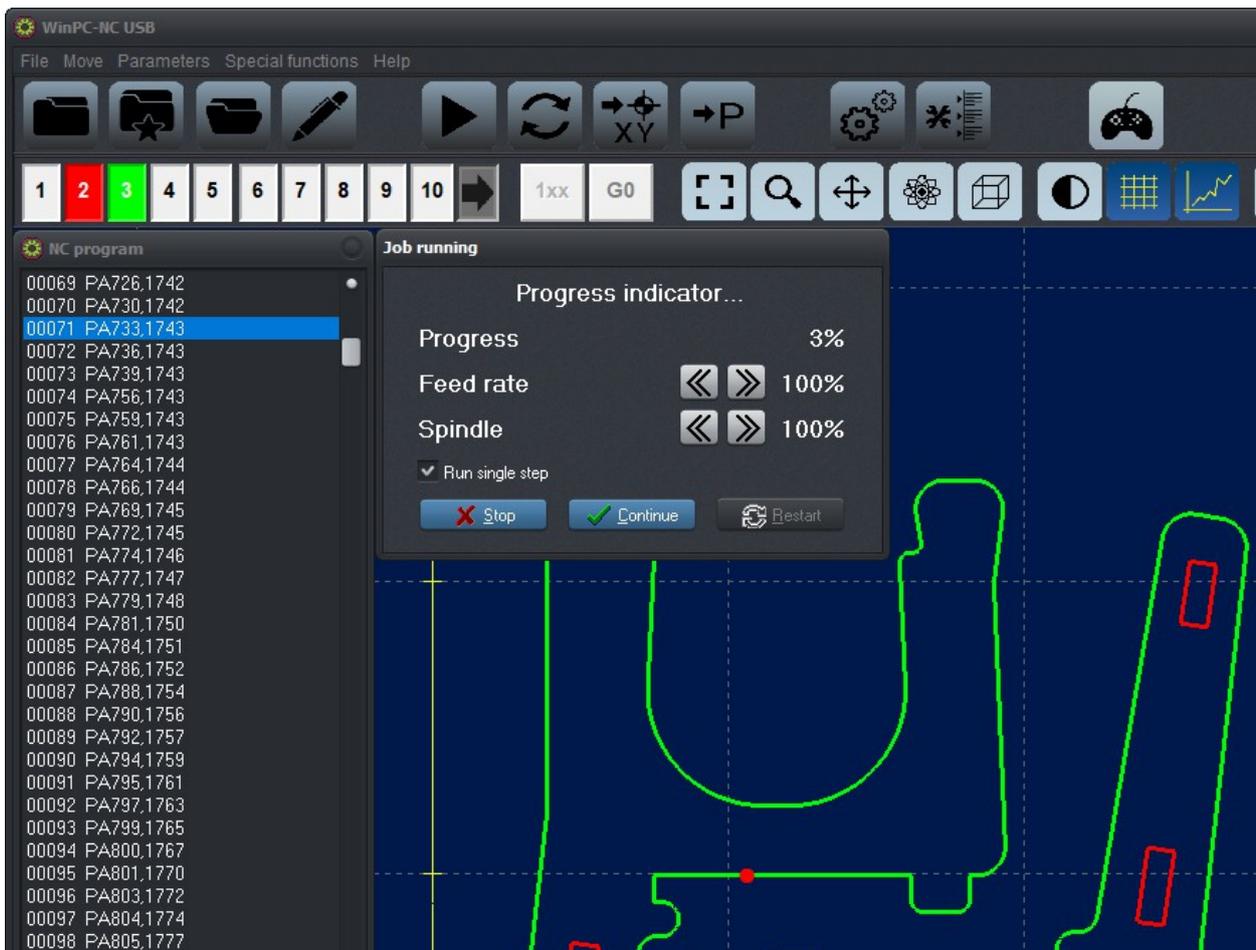
Start a job in single steps and commands

MOVE - ZERO POINT XY and MOVE - PARK

These items cause the machine to move to the named location.

Care should be taken to ensure that a collision does not occur.

The $\boxed{F4}$ and $\boxed{F9}$ keys are accelerators for these locations respectively.

MOVE - JOG

The item enables manual movement of the machine.

Jogging can also be accessed by pressing $\boxed{F5}$ or the JOG button.

JOG function with 3 axis

moving your machine the exact to step by keyboard or mouse

Movement may be performed via numeric key pad or mouse. Step-by-step, precise distance, or continuous motion may be specified.

Briefly pressing a key or clicking the corresponding direction button with the mouse executes a single motion command. Holding a key down or keeping the mouse button pressed causes continuous motion. The changeover time can be defined as a parameter.

Travel speeds can individually be defined to actual requirements. Slow speed is made by keyboard, button kept pressed `CNTL` means fast speed mode. The right arrow keys `1` to `9` move the X and Y axes individually or diagonally, the buttons `+` and `-` move the Z-axis. Speeds are defined in parameters.

Display of step counter   The machine position is displayed in two ways. The first is relative to the the home switches. The second is relative to the zero point of the work piece.

WinPC-NC **maintains two coordinate systems. The first is relative to the machine and is based on the home switches. The second is relative to the zero point of the work piece. The work piece coordinates are often relative to the bottom left-hand corner of the work piece.**

How *WinPC-NC* will move is shown in the lower left hand corner of the jog window.

When continuous is displayed, motion will be in discrete steps. If a distance is displayed, motion will be in discrete distances of that size and units.

The discrete distance may be entered or selected from the pop up menu. The size and units of the menu distances are determined by the units setting in parameters (Metric or Imperial).

As long as a key or button is pressed, the machine will move. As soon as the key or button is release, the machine will come to a halt. No steps will be lost by this process.

Checking the Reset to continuous checkbox will immediately reset the movement to steps (continuous).

Be careful! Jogging can result in a collision which can damage the machine, spindle, or tool.

These spindle and coolant check boxes control their respective functions during jogging.

Closing the Manual movement window ensures that the spindle and coolant pump are turned off.

Zero point XYZ
Zero point XY
Zero point X
Zero point Y
Zero point Z
Zero point 4
Park position
Block - zero point Z
Block - Z depth
Block - Max Z depth
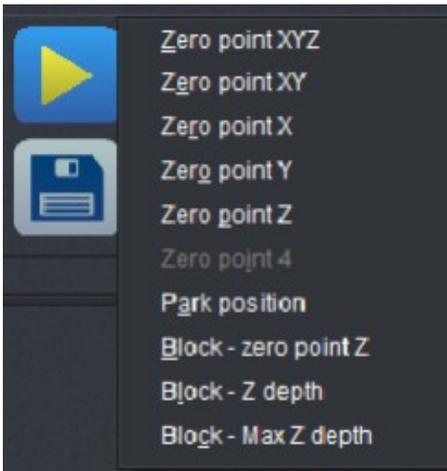
It is possible to move to certain well-known locations. Simply click on the *Move to* button and selection the desired location. The machine will move to location.

Most functions for discrete moving can be accessed by speed buttons shown below.

Zero point XYZ
Zero point XY
Zero point X
Zero point Y
Zero point Z
Zero point 4
Park position
Z depth
Max Z depth
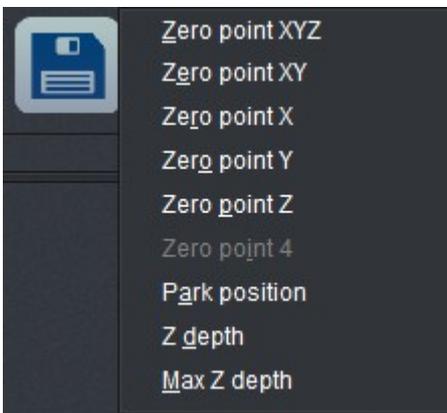
The current position can be saved as one of the well-known locations. Simply, click on the *Save to* button and select the location.

Most functions for saving positions can be accessed by speed buttons shown below.

Savebuttons
(jog window on the left side)

Movebuttons
(jog window on the right side)

A movement in progress can be stopped by clicking the *Stop* button.
Click on the *Exit* button when you're finished jogging.

automatic messurement of Z heights

WinPC-NC can automatically determine certain Z-axis heights via a probe or surface block sensor. The sensor should be connected to an input; its cable must allow easy placement anywhere in your machines work volume.

The sensor height will limit the thickness of the stock that you can place in your machine.

The measuring sequence involves several steps:

- Move the machine over the point to be measured

- Place the probe or sensor on top of the work piece and underneath the tool.

- Start measuring process. ***WinPC-NC*** slowly moves the Z-axis down until the sensor contact trips. Motion stops when the sensor trips and ***WinPC-NC*** uses the stopping position and the height of the sensor (a parameter) to set the Z-axis height.

moving 4th axis

If a 4th axis is available, the appearance of the jog dialog is somewhat different. It includes the expected buttons as well as a control for the rotation. The rotation axis is defined as a parameter. The letter associated with it appears in the dialog.

Jog dialog with 4th axis

*SOFTSTICK –
simulated joy-
stick*

SOFTSTICK is a simulated joystick. The center point is establis-
hed by clicking the left mouse button. Moving the mouse with the
left mouse button held is analogous to moving a joystick.

The jog speed is governed by the three speed selection buttons.
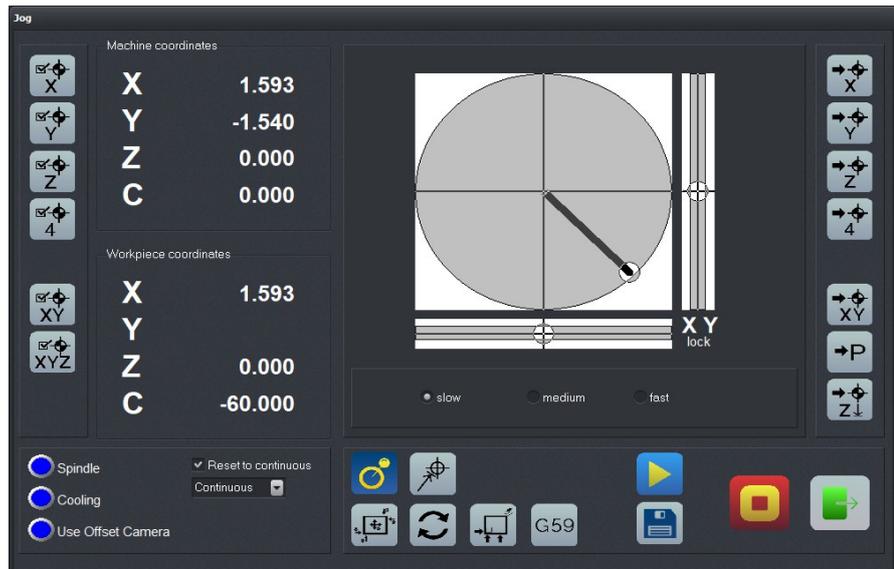
SOFTSTICK function for freely movable axis

Releasing the mouse button halts any motion. Three different
speeds are selectable. The circle controls moving in XY and the
right-hand bar controls moving in the Z-axis.

By clicking the lock symbols the specified axis will be fixed, pre-
venting any further movement.
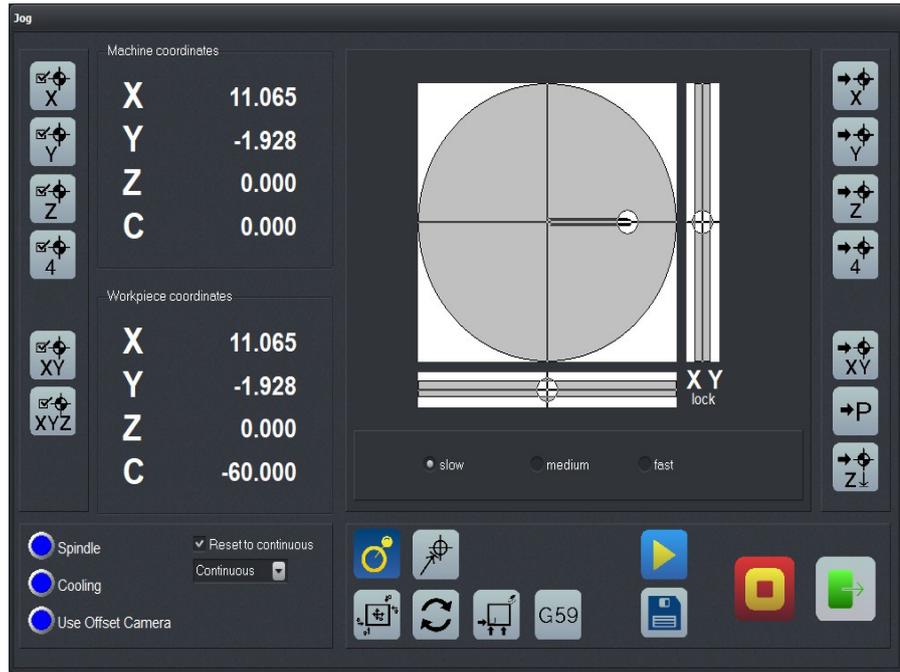
SOFTSTICK – move with fixed X-axis (Lock-X)

Specific movement

Using TARGETED MOVE distances, velocities, and spindle speed can be specified. One can easily perform motions and cuts.

Spindle operation without cooling requires activation of the checkbox *No cooling*.

Specific movement

Edge probe
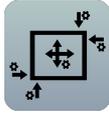
An edge probe automates locating a stock, a work piece edge or the center of a circle.
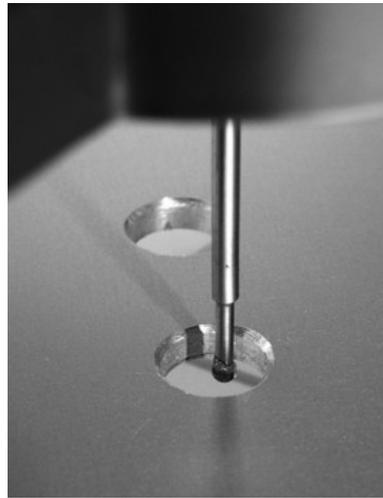
Select the button appropriate to the motion desired. *WinPC-NC* then slowly moves the edge probe laterally until it touches an edge. The motion will stop when edge probe contacts an edge. The edge probe will be backed off slightly from the edge, lifted to the tool lift height, and moved such that the center of the probe is exactly above the edge.

After a measurement, the new position can be saved as zero point of an axis.
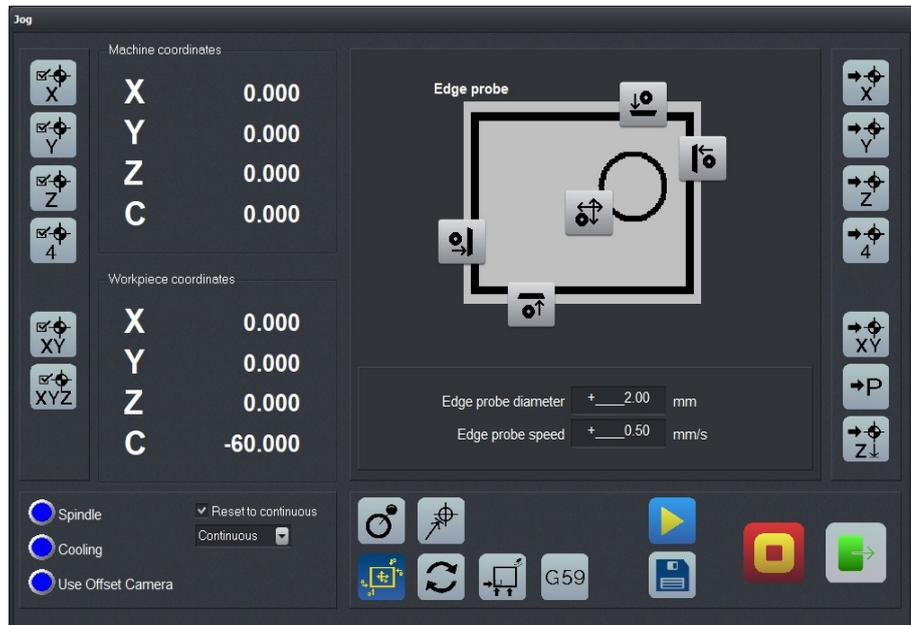
JOG edge probe

To measure the center of a circle *WinPC-NC* moves in the X and Y direction until it establishes four points on the circumference of

the circle. Once the four points have been located, the center of this circle is computed and the probe moves to the exact center of the circle.

Pendant

WinPC-NC supports pendants (hand controllers). These are useful for moving a machine, establishing axis zero points, saving specific location data, and interrupting a job.

WinPC-NC directly supports the **HR-10** pendant. It can be ordered from us. It is a USB device and comes with a 3m cable.

Moving around has never been so easy!

Further more information about the **HR-10,** find the documentation on our web site.

G56

*different zero
points*

WinPC-NC is conceived for operation with multiple zeros (G54 – G59).

This function must be activated in parameters – basic setting – measurements/dimensions and, consequently, in jog move a button is displayed, indicating the current zero.

All functions (move to zero, set zero) refer to this zero point. By pressing this button the coordinates of the different zeros are displayed. Pressing button G5x allows determining another zero point.

We published detailed information on zeros in our homepage www.Lewetz.de, see topic "*Help*" – "*Howto*" – *Instructions*.
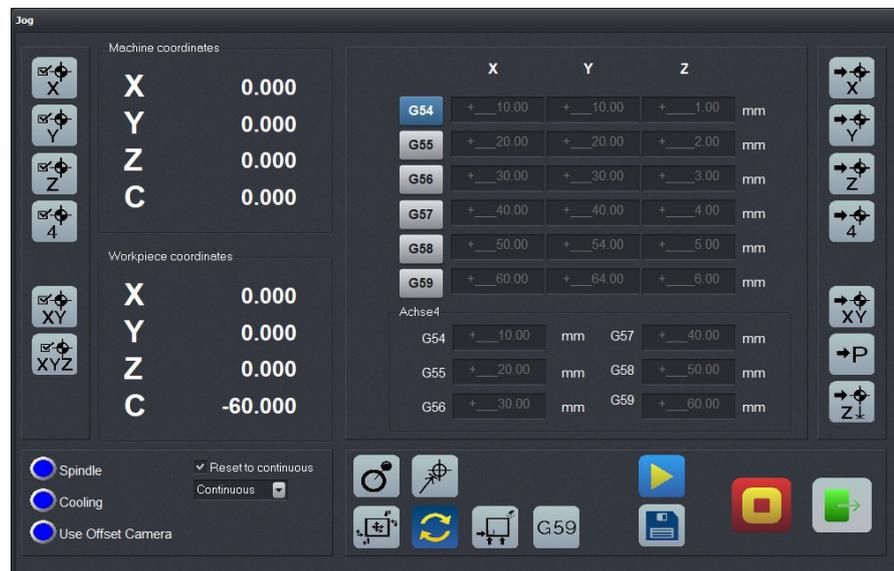
Working with multiple zeropoints (G54..G59)

Contact plate

By means of a contact plate this function allows definition of the zero point of your workpiece.

Additonally it is also possible to execute a phase angle correction, if the workpiece is not exactly fixed with its edges parallel to the machine axes.

Ready for download are detailed instructions in our website www.Lewetz.de, see topic "*Help*" – *"Howto"*.
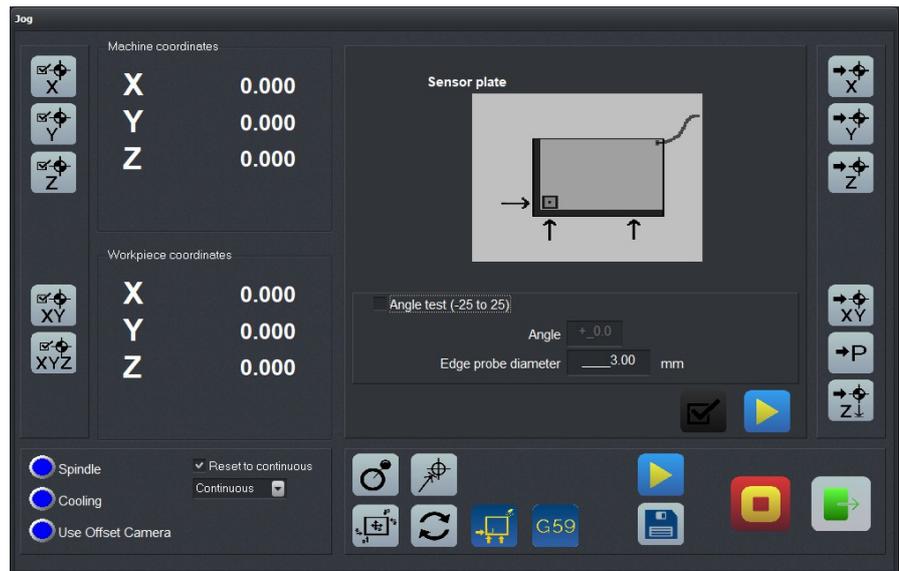
Using a contact plate with WinPC-NC

MOVE - JOYSTICK JOGGING

A joystick - rather than using a mouse, a keyboard, or a pendant - can be used for jogging.

Prior to use, a joystick must be configured in Windows system and calibrated in *WinPC-NC*.

Joystick jogging

To move in X and Y, simply move the joystick. To move the Z-axis press and hold button 1 while moving the joystick. A help point can also be stored while using the joystick. To save the help point, press button 2 at the same time as holding button 1.

Display jog with joystick

Space mouse 3DConnexion

The 3D space mouse from the 3DConnexion company is supported. They make moving in 3 or 4 axes highly intuitive.

Space-Mouse-Wireless from 3DConnexion

Before using one, install the latest Windows drivers. Define the type of connection the device using in the parameter menu *ports*.

MOVE - INITIALIZE MACHINE

Before a machine can be used, it must be initialized. The initialization procedure establishes the reference positions of the machine - the points the machine will use as X0, Y0, and Z0.

*Initialize
the machine*

The machine cycles through each axis, slowly moving until the respective home switch trips. It then moves back slowly until the home switch ceases tripping. This is the reference position for that axis and can be defined as zero point or as any different axis coordinate.

WinPC-NC needs to know exact machine positions. Therefore an initialize run should be performed at each machine and program start up.

Machine initialization can also be performed by pressing ⌈F8⌉ or the initialize button.
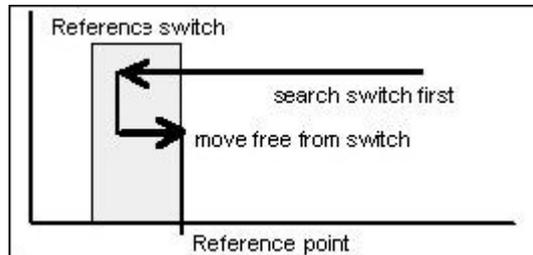
Moving axis to home position and switch

MOVE - SELECT TOOL

This item brings up a dialog box for manipulating tools. It can also be activated via the Tools button.

The tool dialog box can:

• Load and unload an ATC tool

• Pick up a new tool

• Measure the length of a tool

• Identify the tool which has just been inserted

As a rule, *WinPC-NC* always remembers which tool was last used and the status of the collet chuck. This information is retained from session to session.

ℹ In order to be able to use each function of this dialog at least one tool length sensor should be set up.

Using an Automatic Tool Changer (ATC) requires the corresponding activation in the parameter functions, an ATC magazine, and a programmable, powered collet chuck.
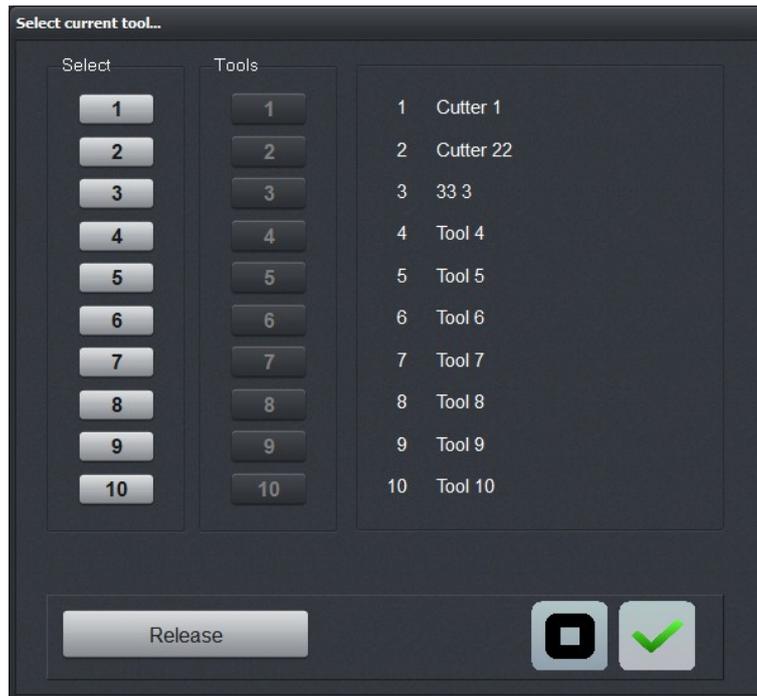
Dialog box for selecting, picking up, putting down, and measuring tools

Functions of buttons:

• Release returns the tool currently in the chuck and returns it to the magazine

• The *molett* button opens or closes or opens the chuck. There is a security prompt prior to opening it

• Stop cancels any motions (e.g. tool length measurement or moving to a magazine location)

• OK closes the dialog box

To inform *WinPC-NC* which tool is currently inserted, click the tool number or its name on the right. This may be necessary after the initial startup or following a cancel.

WinPC-NC automatically handle tool length compensation issues

tool length compensation

Follow this procedure:

1. Insert a tool – the one you want to use first.

2. Load your job if it isn't already.

3. Click on the tool number associated with the tool. The tool number is immediately displayed in the status bar. *WinPC-NC* then moves the tool to the length sensor and takes a measurement. The length of the specified tool is saved and displayed.

4. Load the job stock and define the job's X, Y, and Z zero points. The Z zero point can be established by carefully lowering the tool until it barely scratches the surface of the stock.

5. Start the job. As each new tool is required, *WinPC-NC* will measure the tool length automatically and make any necessary adjustments.

   WinPC-NC calculates the exact length differences to the master tool and compensates it during the following job process. There are no changes in saved zero points or Z heights.

3.3.3. PARAMETERS menu

The PARAMETER menu contains all of the *WinPC-NC* settings and options. The settings and options are handled via several dialog boxes, grouped by function.

```
Coordinates
Tools
Miscellanous
Technology
Data format
Basic settings

Save profile
Selectively save profile
Load profile

Save
Save as...
Load
```
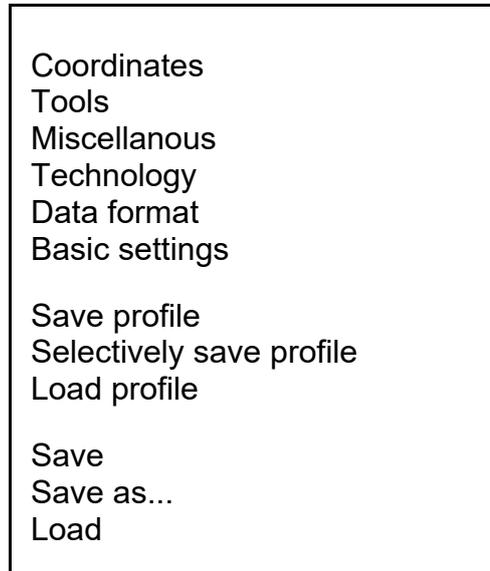
Pressing Alt-P opsen the parameters menu.

Clicking the PARAMETERS button opens the parameters dialog immediately.

A specific summary and description of parameter settings can be found in a later chapter.

PARAMETERS - SAVE/LOAD PROFILE

Saving of complete profiles includig all settings and macros

WinPC-NC offers a function to save and reload complete profile settings for different applications like milling, tangential cutting or 3D printing. The saved setup file includes all parameter settings of *WinPC-NC* itself and of the machine as well as defined macros and more settings.

Next to a system changeover all applications can easily be activated by selecting the corresponding setup file and the software can be properly configured. .
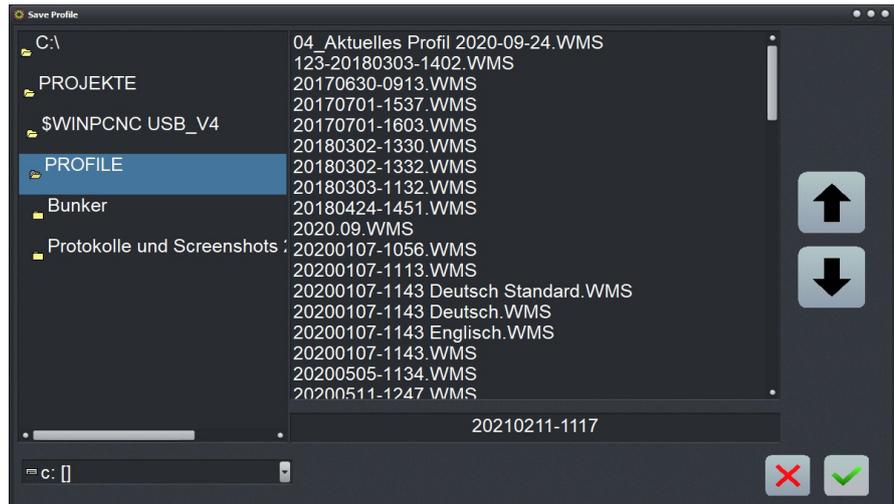
Profile save dialog

Files with profiles (machine-set-up) are marked with *.WMS from **WinPC-NC USB** and *.WPMS from **WinPC-NC Professional**. They comprise any settings concerning the machine, WinPC-NC, tool settings, macros, messages and additional characteristics.

Loading a complete profile (Setup)

After refitting the system, it is easy to activate all applications by selecting the specific setup file (Profile) and to install the software.

In **WinPC-NC Professional** files with the machine setup are marked with *.WMS or *WPMS and they comprise all machine settings, even for **WinPC-NC** itself, all tool settings, macros, messages und additional characteristics.
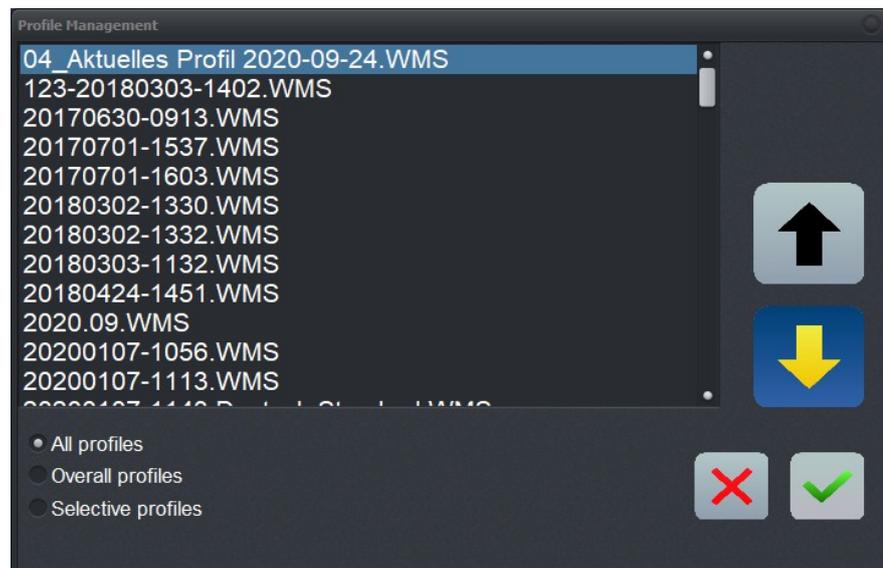
Profile management

PARAMETERS - SELECTIVELY SAVE PROFILE

Selective saving of a profile
As some parameters apply to the entire machine and others are only specific to a particular component or accessory, these parameters can be saved separately in a profile.

Saving only few selected parameters
For this purpose a template file is used for defining parameters to be saved. These files are stored in the sub-directory *TEMPLATE.* If such *a* template file exists, it is displayed and can be used for saving defined parameters.

After confirming with the OK button, it is possible to specify term and address for the selective file. For saving this data click OK again. In **WinPC-NC Professional** the selective profile files end with *WMTS or *WPMTS
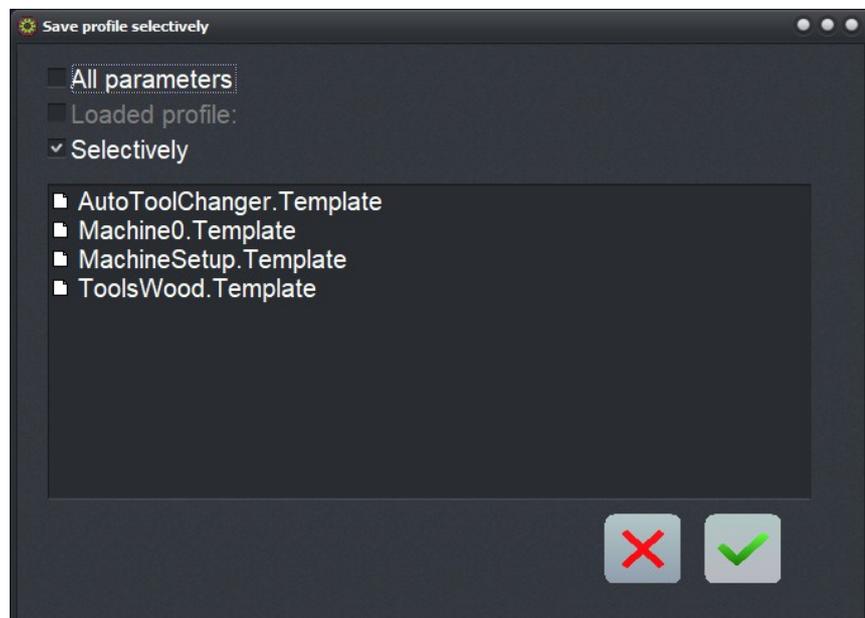
Selectively save profile

PARAMETERS - SAVE

save parameters to a project or job
This item saves all of the parameters and tool settings associated with the currently loaded NC file.

All of the work piece related settings (e.g. zero point, scaling) are saved in parameter and tool files associated with the working file name. These files have the same name as the working file, but with extensions *.WPI and *.WPW

When the NC file is loaded in the future, all the settings and tools information is restored.

The machine-related parameters (e.g. axis resolution, reference directions, interfaces used) are always stored in the standard parameter file WINPCNC.WPI.

If no working file is selected when you save, the settings are saved into the WINPCNC.WPI and WINPCNC.WPW files.
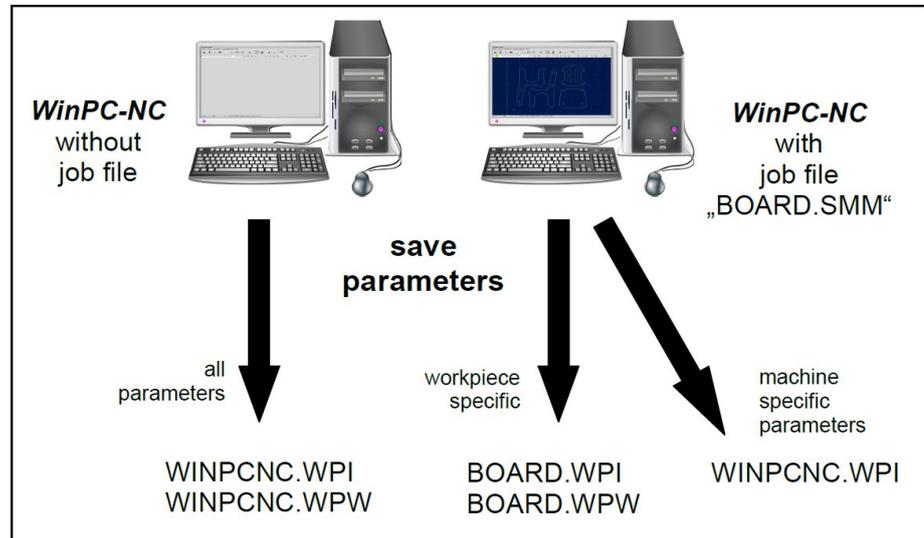
Saving parameters and tool settings

Separating the parameter files offers the advantage that the machine parameters will probably only have to be saved once; other settings only have to be saved if something changes.

PARAMETERS - SAVE AS...

This item allows you to enter the name for the project parameters file. Using names appropriate to a procedure or material is a good way to handle similar jobs in the future.

This function saves project related settings only but not any system or machine related settings.

Avoid overwriting the standard parameter file WINPCNC.WPI by the using PARAMETERS-SAVE AS....

PARAMETERS - LOAD...

load settings for specific applications or jobs   The LOAD function enables you to load various specific parameters for different materials or working processes.

A window appears with the familiar open dialog box functions. It is possible to select parameter or tool files.

The PARAMETERS-LOAD function can also be activated with `Shift-F2`.

3.3.4.   SPECIAL FUNCTIONS menu

The SPECIAL FUNCTIONS menu provides access to a variety of functions useful to test, optimization, debugging, and calibration.

```
Signal test
Motor test
2D-CAM
Laser grayscale test
Laser cutting test
Controller information
Calibrate joystick
Rehoming check
Factory settings
Digitizing
Teachin
Spindle warm up
```

SPECIAL FUNCTIONS - SIGNAL TEST

This item displays the status of many of the critical input and output signals (e.g. home and limit switches).

test input and output signals interactively

WinPC-NC continuously monitors its input and output signals and displays a status of a subset of them here. This is useful for testing.

Grayed out signals have either not been defined or do not have an input/output pin assigned to them.

Black indicates the switch is not active, red indicates that the switch is activated.

Use PARAMETERS-BASIC SETTINGS-SIGNAL WIZARD and SPECIAL FUNCTIONS – SIGNAL TEST to establish and test the function to signal assignments for all inputs and outputs.
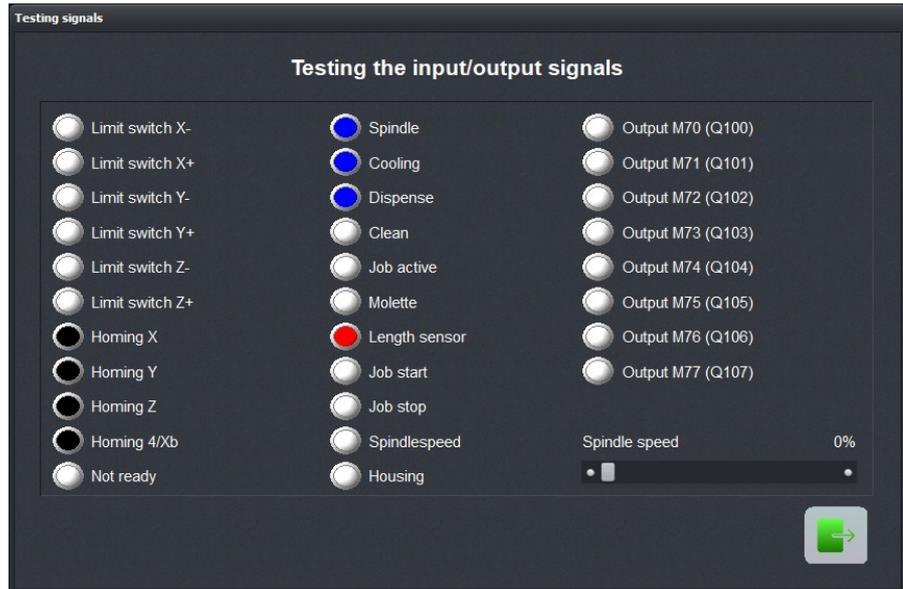
Signal test

Output signals can be tested by simply clicking on their LED symbols. Blue indicates on and green indicates off.

testing spindle speed control The slider at the bottom right-hand corner of the window is used to test spindle speed control. Depending on how **WinPC-NC** is configured, the slider will either vary an analog output (0-10V) or a PWM signal.

WinPC-NC Professional can handle a real analog output signal with 0-10VDC depending on the configuration of the axes controller.

SPECIAL FUNCTIONS - MOTOR TEST

This item is used for determining the optimum speed settings. The *Testing speed* window displays all of the parameters relevant to a stepper speed evaluation.
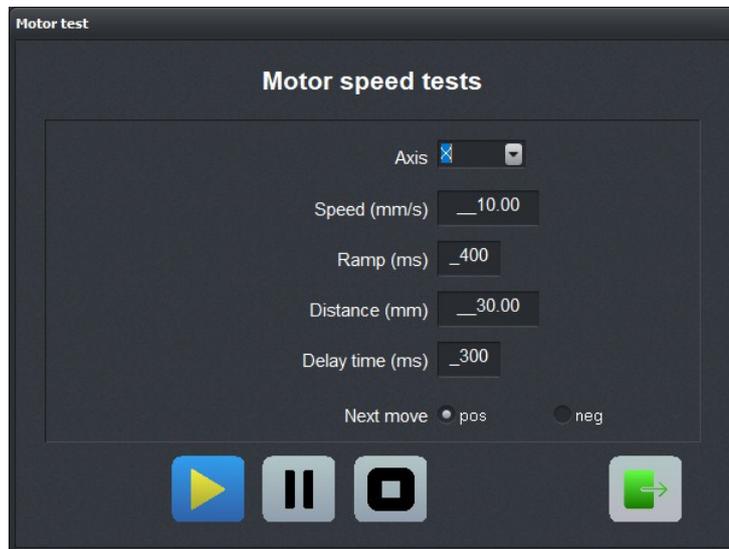
Motor test

Enter the parameters you wish to test and hit the *Move* button.

WinPC-NC will move the selected axis forwards and backwards continuously. By listening to the machine and observing the movement, it is easy to tell whether the parameters are valid for the axis, or whether additional corrections are needed to the speed or ramp length.

The test run can be cancelled by pressing ESC or clicking the *Stop* button.

Optimum parameters

The optimum values for an axis have been achieved if the motor starts up quickly without step losses, and is still able to develop a sufficient amount of torque when running at maximum speed.

best configurations

Step-by-step procedure for testing X, Y, and Z-axis

1. Switch off the ramp length and slowly increase the start/stop speed until the motor stalls. Reduce the speed value by 30-40%.

2. Test various ramp lengths. A good value is one where the motor starts up quickly and does not stall.

3. Increase the rapid speed **slowly**. The motor should run quickly while still developing a sufficient amount of torque.

Once you're comfortable with the parameters, save them.

All related settings are mentioned in later chapter of parameters.

The optimum parameters for stepper motors depends on many factors, including the motor characteristics, the type of drive used (direct or belt), and the load being driven.

It is _impossible_ to read machine and motor specifications and determine an optimal set of parameters. If you are unsure of how to tune a machine, stay with the factory provided defaults.

SPECIAL FUNCTIONS - 2D CAM

The setting dialog concerning contour optimization, automatic computation of radii correction and starting path opens. Detailed information is available in point 4.

SPECIAL FUNCTIONS - CONTROLLER INFORMATION

This item will identify the external module as well as display important information about its characteristics as well as different running and time counters.

Before reporting any issues - hardware or software - bring up this information, take a screen shot of it, and send the screen shot along with your questions.

Controller information

SPECIAL FUNCTIONS - CALIBRATE JOYSTICK

This item performs joystick calibration

easy calibration of a connected joystick   Instructions for operating the joystick are displayed in a window. Please follow the procedure exactly.

A joystick cannot be used until it is calibrated.

Joystick calibration

SPECIAL FUNCTIONS - REHOMING CHECK

This item is used for checking the accuracy of the reference
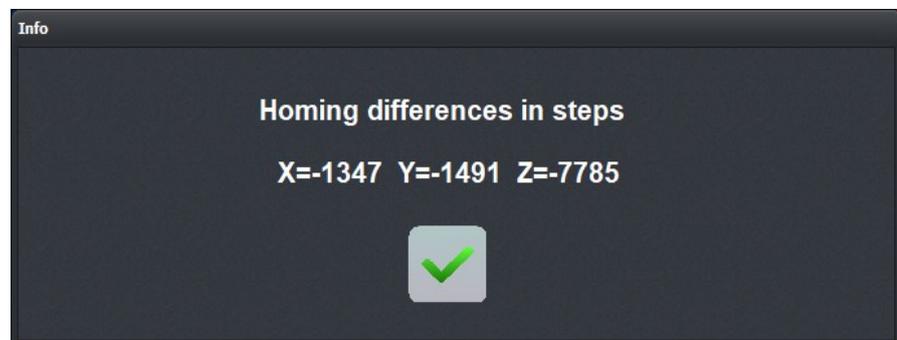
Result of the rehoming check

contolling of initialisation

Relatively small step discrepancies may be due to the switches. Large discrepancies, on the other hand, indicate step losses. Step losses indicate a serious problem.

Use SPECIAL FUNCTION - CHECK POSITION if:

- A collision has occurred - steps may have been lost.

- You want to determine the maximum machining speed for a tool or material, and you're concerned that loss of stepping is occurring.

- The position was changed during a tool change.

Check position can be performed only if the machine was initialized. The initialization procedure must have completed without errors (no limit switch or stop signal issues occured).

WinPC-NC **cannot initialize a machine or check position properly unless the home switches have been properly installed. The home switches must be installed a small distance from the mechanical edges of each axis such that the initialization procedure can cause each switch to trip and then back away until the switch ceases to trip.**

SPECIAL FUNCTIONS - FACTORY SETTINGS

This item restores *WinPC-NC* to its original factory settings. Any settings made previous will be lost.

If *WinPC-NC* came bundled with your machine, you will be prompted to identify your machine. This will reinstate the necessary parameters for your machine.

If your machine was provided with a set of parameter files (e.g. USB key or emailed files), you will want to locate them and load the parameters again.

SPECIAL FUNCTIONS - TEACHIN

The TEACHIN function makes it easy to create simple programs in various formats by moving and following the contours.
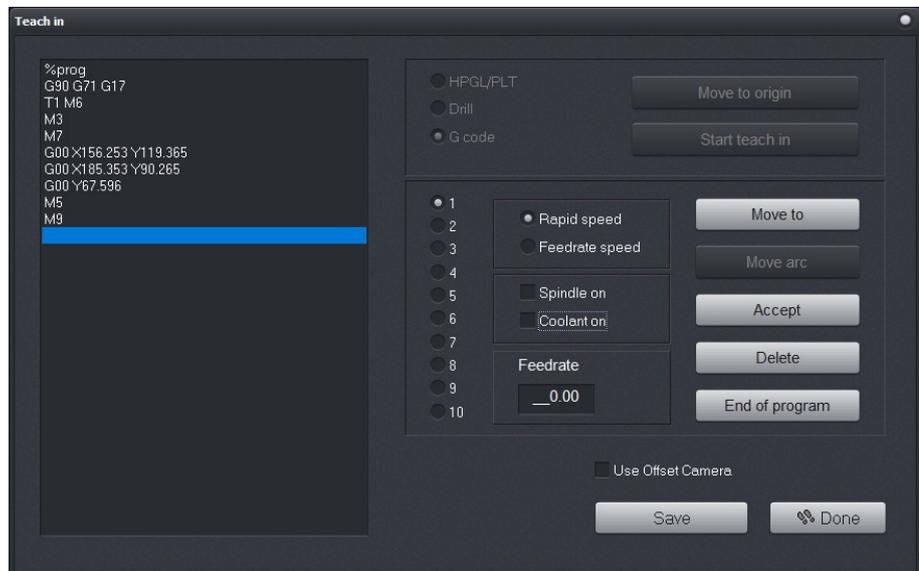
Special functions-Teachin

create
nc programs
interactively

The new program is displayed in the Teachin window on the left. The top right displays those selection that must be chosen before creating an NC program.

The bottom right displays all available actions (e.g. tool selection, speed settings, and spindle and cooling control).

The choice of acceptable commands depends on the selected data format. For instance, spindle and cooling command are not handled in the HPGL format.

Teachin step by step	**Suggested procedure with Teachin:**

1. Execute a reference move and call up the Teachin function.

2. Specify the data format and zero point (by jogging).

3. Start the Teachin process. The program header will automatically be set up; it will appear at the top of the program area.

4. Enter the necessary commands. Care should be taken to ensure that the command sequence is correct and the correct tool is used by the expected command. You can define straight lines or arcs or ongoing contours.

5. Click on exit button and the program is automatically finished.

6. The new created program should be stored before leaving Teachin.

Possible actions during the Teachin process:

Movement in a line	Choose the movement speed, either the rapid rate (fast movements) or the feed (cutting) rate.
Movement in a circlular arc	A circle or arc always requires three points. The first point is the current position. The other two must be points on the circle/arc. From three points a unique arc command can be created.
Accept	This function inserts the actual position of the cursor bar into the program.
Delete	Deletes the line under the cursor bar.
Exit	Inserts the necessary commands for exiting the program and finishes the new created NC program.

New commands are always inserted at the actual cursor position. This enables the user to make up for overlooked actions.

SPECIAL FUNCTIONS - DIGITIZING

Scanning surfaces

The function *DIGITIZATION* is a powerful tool for copying existent workpieces. However, this calls for an integrated sensor or sensor pin, which approaches the workpiece from the top and switches on contact or triggers level change on signal button I221.

Inserts the required commands for program end and finishes the new created NC program.
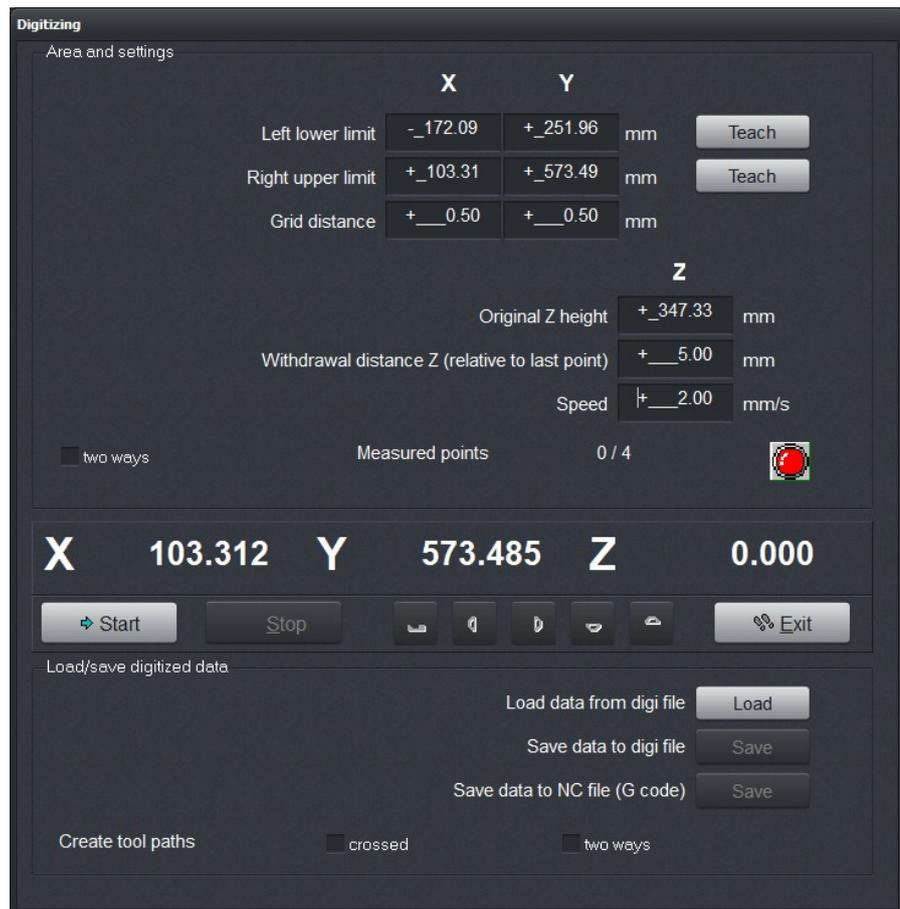
Special functions-Digitizing

Elements in the window

The window of DIGITIZATION is made up of two different components. All entries for specifying measuring area and measuring method are to be made in the top area. Processing and saving the data is made in the lower area after the measuring process.

Lower left corner
Top right conrer
Defines a rectangle in absolute machine coordinates where surface measuring is made. Both corners can be easily approached by jog move.

Measuring distance
The measuring distance defines the distance between two sequential series of measurements for both axes.

Initial height Z
Defines the first Z-height where measurement starts in the lower left corner.

Retraction height Z
Defines the distance upwards of the Z-axis after measurement for approaching the next measuring position   .

Measuring point X/Y
Indicates during the measuring process the total number of measuring points which will be approached and the number of those which are already measured.

LED
As test display indicates the status of the measuring sensor and it is easy to execute manually a functional test. The LED colour must be black, by pressing the colour must be red

Forward/Reverse
Defines measuring direction: bi-directional or uni-directional. Machines with backlash compensation should be measured uni-directionally.

Start/Stop
Starts or stop

Digitising a model

Surface measuring with defined limits

Measuring starts in the lower left corner of the measuring area and scans the model line by line in accordance to the pre-de-fined distance. Operation is always made along the X axis and slowly approaches the end position in the top right corner. The current positions of the axes are displayed online in the position area.

After measuring the data can be processed by multiple functions. Prior to any other operational step, it is recommended to save the current data as measured-value file. In case of faulty operation this file can be reloaded again. The measured-value file is saved and opened by the relevant buttons and the desired file name.

For correcting measured values following function keys are available:

Creates a negative of the digitized model, i. e. an original will be converted into a form or vice versa

Copies the model, mirrors it to the right corner and connects it seamlessly.

Copies the model, mirrors it to the left corner and connects it seamlessly

⊖	Copies the model, mirrors it to the upper corner and connects it seamlessly
⊖	Copies the model, mirrors it to the lower corner and connects it seamlessly

ⓘ **Copying, mirroring and attaching is particularly recommended with symmetric parts in order to minimize digitization.**

Saving measured data as operational CNC program is made in the final step and data output can be affected by two additional setting options.

Crosswise *WinPC-NC* is induced to begin milling along the X-axis and then, in the second session, along the Y-axis. By means of crosswise smoothing the surface becomes a clean finish.

Forwards/ Backwards Defines milling mode: bi-directional or uni-directional. For machines with backlash on reversal we recommend uni-directional mode.
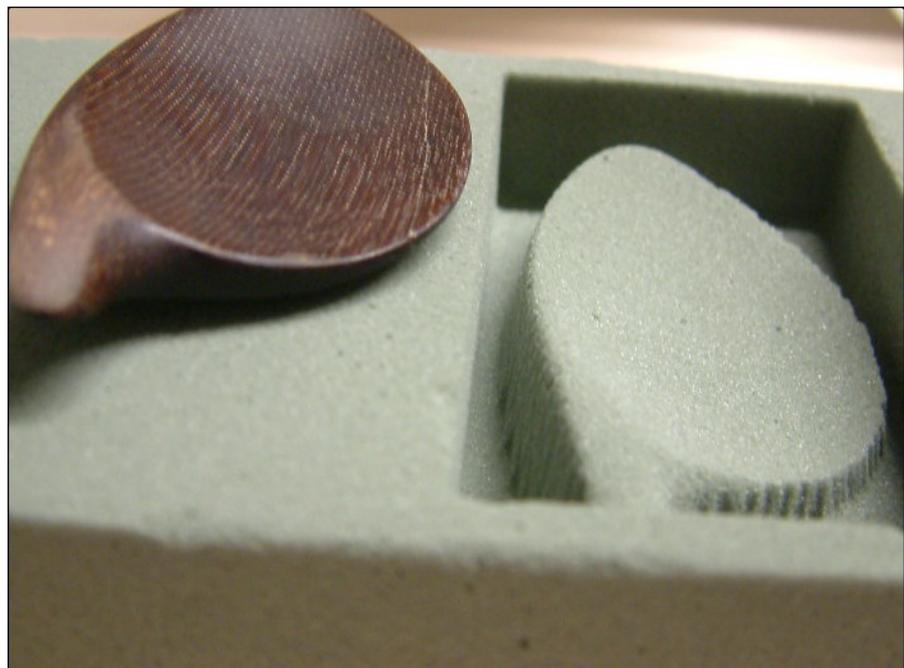
Digitised model and reproduced copy

SPECIAL FUNCTIONS - LASER GRAYSCALE TEST

For detailed information and technical instructions concerning the correct use of the laser please visit our homepage at www.Lewetz.de. By a click either to the "*Help* menu – Howto" or „Configure laser and laser test functions", you will receive all relevant information

The grey scaling test is subdivided in two sections: first section for optimum speed, second section for grey gradation.
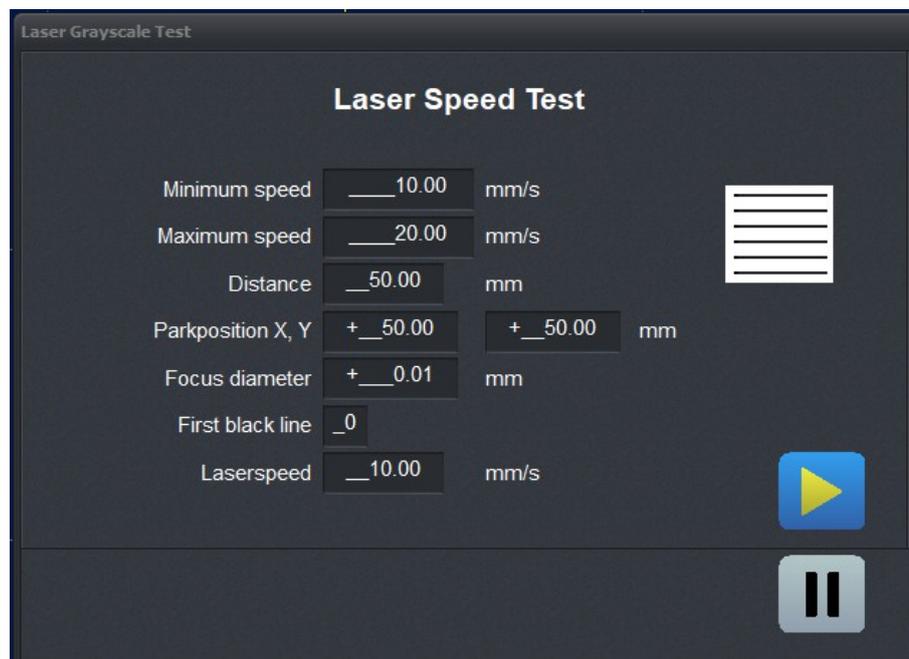
Part 1: Determination of fastest possible speed

1. Laser speed test

First the maximum speeds for a very deep black colour can be determined by the parameters at the left side of the window. Therefore the laser operates with different speeds at maximum power. Then you select the most suitable line being generated in a deep black. By selecting the line in the dialog box, speed can be automatically saved and, in the next step, it is easy to execute the grey scale matching

For the first test run we recommend using standard parameters as illustrated. If speed was too low or too excessive, parameters can be customized and the test can be repeated.

For starting the speed test, please move to the zero point of the sample by using the JOG move function, which is available in the laser test by the key at the bottom left.

Subsequently please save your XY zero point and correctly set up the focus height of the laser.

Now all basic requirements are fulfilled for launching the speed test which can be made by clicking the *Start* button on the left side of the window.

WARNING. Your machine starts now automatically. The test can be interrupted by the *Stop* button at any moment.

In order to be able to display grey shades with the laser, it is necessary to adapt the power characteristics of the laser diode to the linear characteristics of the grey scales. For calibrating the correct parameters, *WinPC-NC* provides a grey scaling test.

Part 2: Determing grey scaling with three support points

*2. Laser
grey scaling
test*

Thereby single lines, always offset to each other by 0,1 mm, are traversed according to the previously determined laser speed. For each line power is dimmed gradually from OFF to 100 %. In the case of a successful test run it should be possible to create a linear shape from white to black.

WinPC-NC starts the grey scaling test after activating the *Start* button. The test is made across the lines according to the speed test.
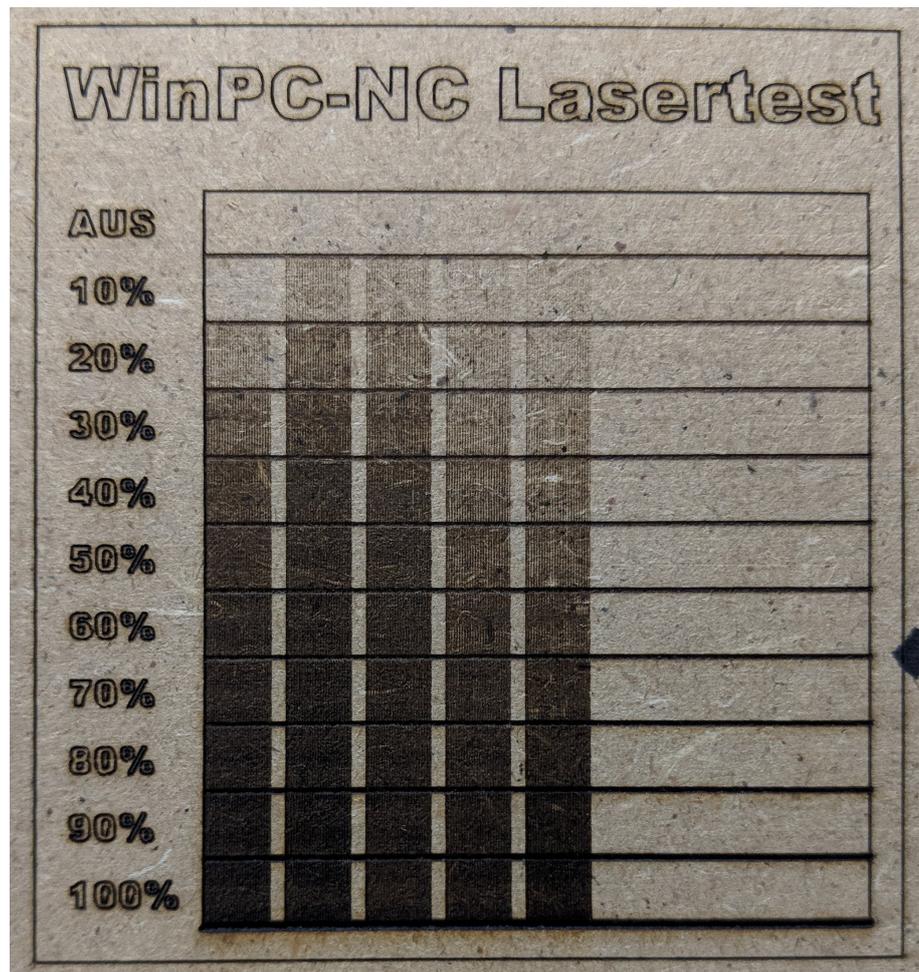
Result: Visible gradations after repeated adaptations of support points

SPECIAL FUNCTIONS - LASER CUT TEST

We provide detailed information and technical instructions concerning the correct use of the laser in our homepage www.Lewetz.de in the category *"Help* menu – HowTo".

*Laser
cutting test*

Some materials tend to melt at the edges. In order to avoid these deformations, it is recommended to determine and set parameters previously by a cutting test.

Hereby the optimum cutting speed and the required number of cuts can be determined with test runs.
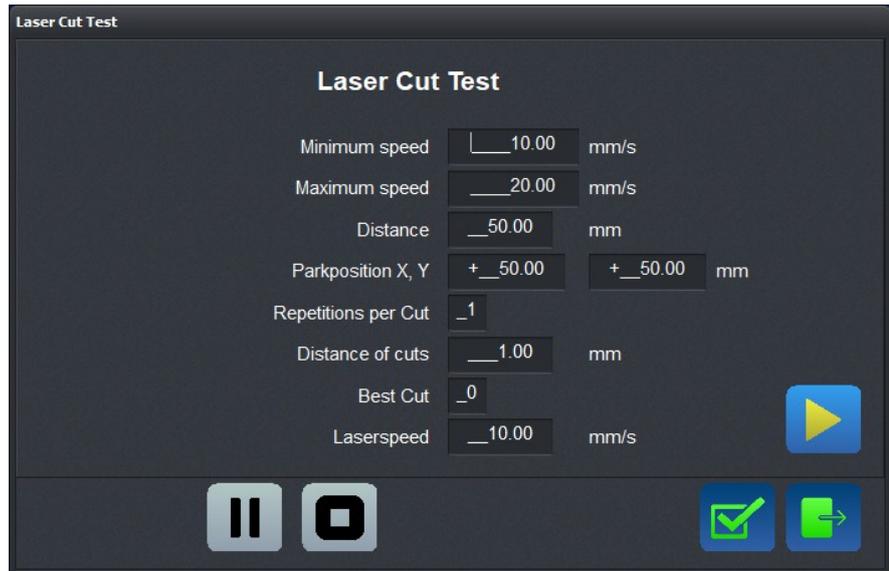
Laser cutting test for determining the optimum speed and sessions

Result: Foil cut with optimum cutting speed

SPECIAL FUNCTIONS - SPINDLE WARMUP

Warm-up time of spindle

By using this function the spindle can be slowly warmed-up by defined speed steps

It is possible to define 5 different speed steps and specify the individual operation time for each speed step.

It is also permissible to specify individual dead times between the speed steps. It is not obligatory to make use of all 5 speed steps which are available.
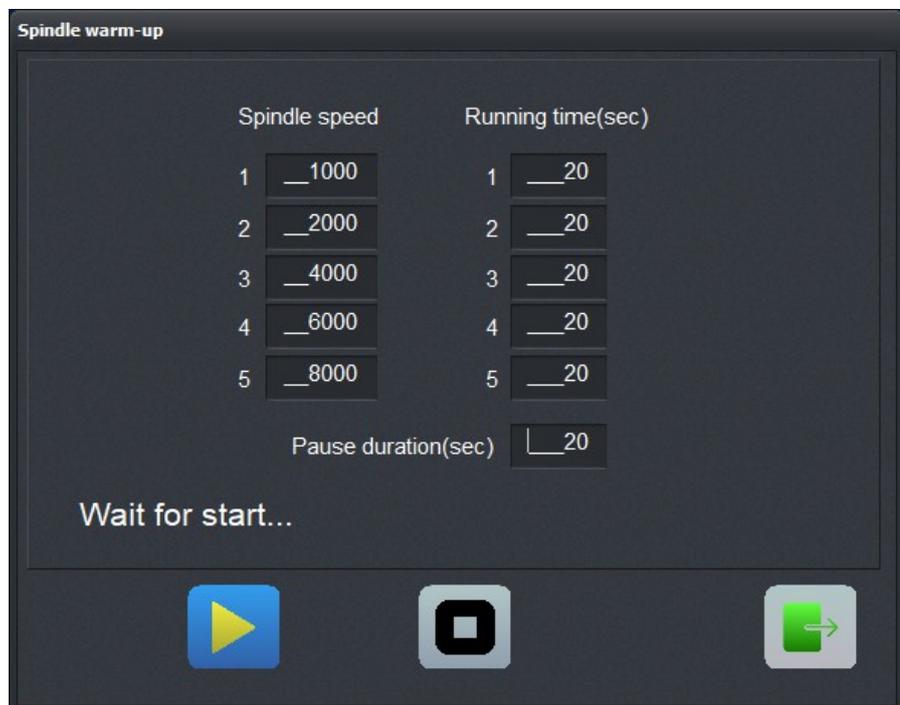
Warm-up function of spindle

By this function **WinPC-NC** defines all selected and specified speed steps according to the pre-defined chronological order and operation times. After the process is finished the spindle stops again.

3.3.5.   HELP menu

There are three items in the help menu:

This is new
Manual
Liability
About *WinPC-NC*
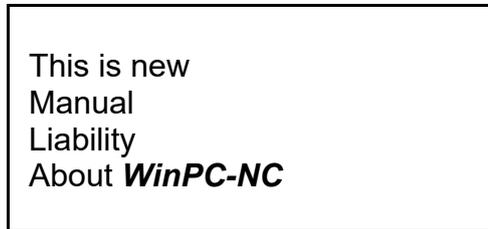
HELP - THIS IS NEW

Each software update comprises a PDF file describing all technical innovations and new features. This file is loaded in the pdf viewer in the *Help* menu – *This is new*.

HELP - MANUAL

Brings up the PDF version of manual in your PDF reader.

HELP - LIABILITY

The HELP-LIABILITY function displays a text with the license conditions and a disclaimer. Please read this information carefully before using *WinPC-NC*.

HELP - ABOUT WinPC-NC

Activating this menu function displays information about the current version and revision number.

Information about the current version

Context-related help can always be summoned via $\boxed{\text{F1}}$. Help concerning the current operation will be displayed.

The ONLINE-Help will bring up this manual. The manual is included in the *WinPC-NC* installation and can be read via a PDF reader (e.g. Acrobat Reader, Foxit Reader).
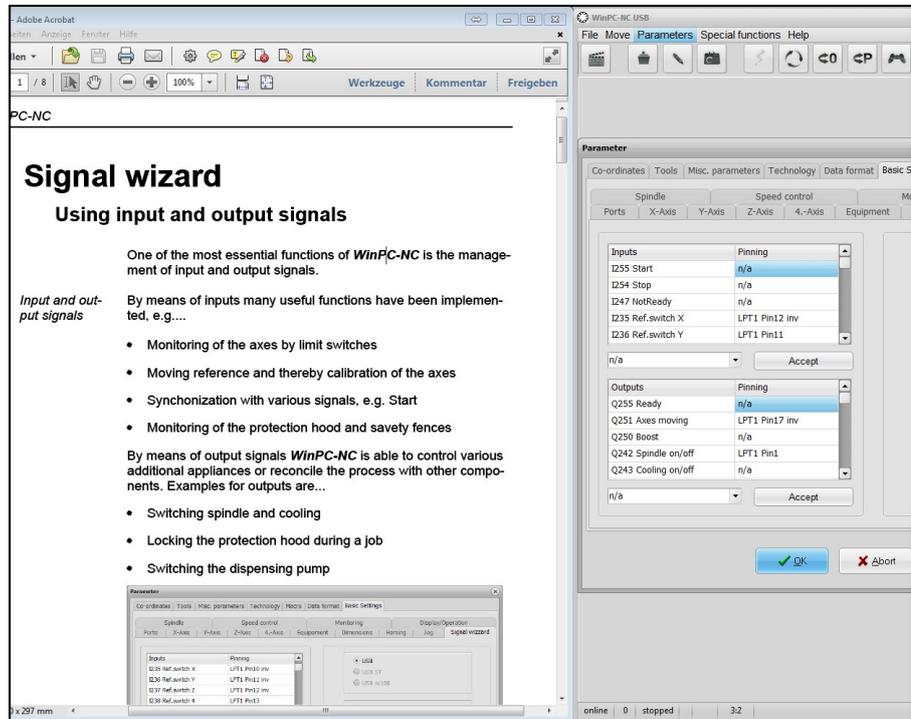
Online help - parameters signal wizard

4. 2D-CAM Functions

4.1. Overview

WinPC-NC provides special functions for sorting and preparing 2D data for subsequent processing. Following functions are available:

- Assigning tool usage

- Cleaning up data, deleting zero length vectors and duplicate lines

- Setting the output sequence according to tool number

- Optimizing empty moves

- finding and sorting ongoing and subsequent contours

- creating moving in vectors to contours

- compensate and recalculate tool diameters

- calculate offsets for dragging knifes when cutting foils

For all
2D data

The 2D-CAM functions are applicable to any imported 2D data (e.g. HPGL, EPS/AI, DXF (2D) and drilling formats).

The CAM functions are activated by clicking in the 2D-CAM button after loading the job. A dialog box will appear. It provides all available options and functions.

4.2. Setting options
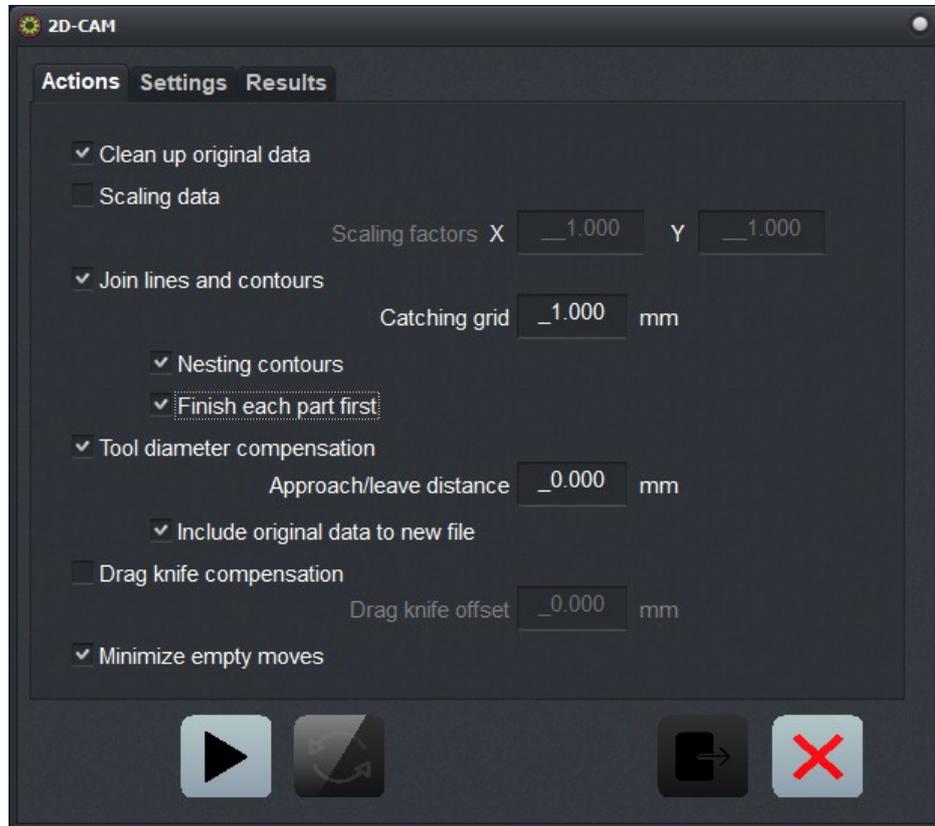
Setting options - 2D CAM Actions

Available functions and settings:

Cleaning up data

Zero length and duplicate vectors are removed from the drawing. These modifications are usually safe and no visible changes will appear. Be aware that these changes may result in subsequent calculation failures.

Scaling data

All data and lines are scaled in size prior to any subsequent calculations. This is an es-sential feature for using the CAM functions as a subsequent consideration of the tool radius for the required target size must always be observed. If scaling is made afterwards via *WinPC-NC* parameter settings, the tool radius offset is also scaled.

Different settings can be made for the X and Y axes.

Searching contours/lines

WinPC-NC tries to create closed contours and continuous, extended lines out of many individual vectors. Individual vectors are sorted and linked. During the search process, consecutive lines may be discovered that do not touch each other end to end – there is a small gap in the data. When a gap is detected – and only when a gap is detected – "fuzzy logic" will be attempted find a close contour and connect the ends.

During the search process, *WinPC-NC* will find all lines, closed contours, and contours which are contained within other contours. Each discovered line and contour will clearly be marked. This is an essential feature for the radius correction later on.

Finish each part first

Flame cutting application is especially used to complete individual pieces. Setting this option means that the piece is cutted from the inside to the outside and then – completely finished - ready for being removed. The next piece is not processed before the previous one has been finished.

Otherwise cuttings are firstly made inside and then the outside contour is processed. This refers solely to the chronological order of cuts during flame and plasma cutting.

Drag knife compensation

This option allows the compensation of the drag knife with definable Offset.
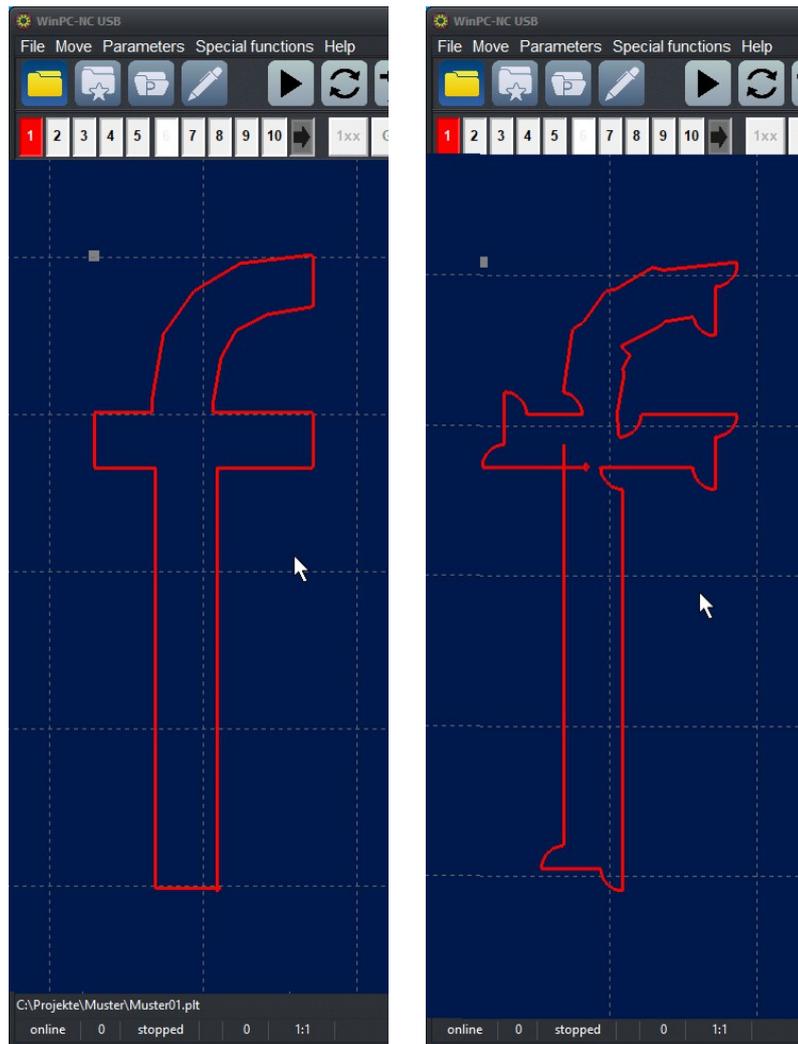
Original contour with calculated drag knife offset.

Calculating tool diameter offsets

The radii of each tool involved is used to calculate an offset appropriate to each contour it is associated with. This ensures that each contour will be precisely milled.

When a correction is made by hitting the *START* button, the tool number of the correction with have 100 added to it. This makes contours that have been already processed and those that require further processing easy to spot.

You also can force *WinPC-NC* to create a moving in and leave vector to avoid start and end directly at contour.
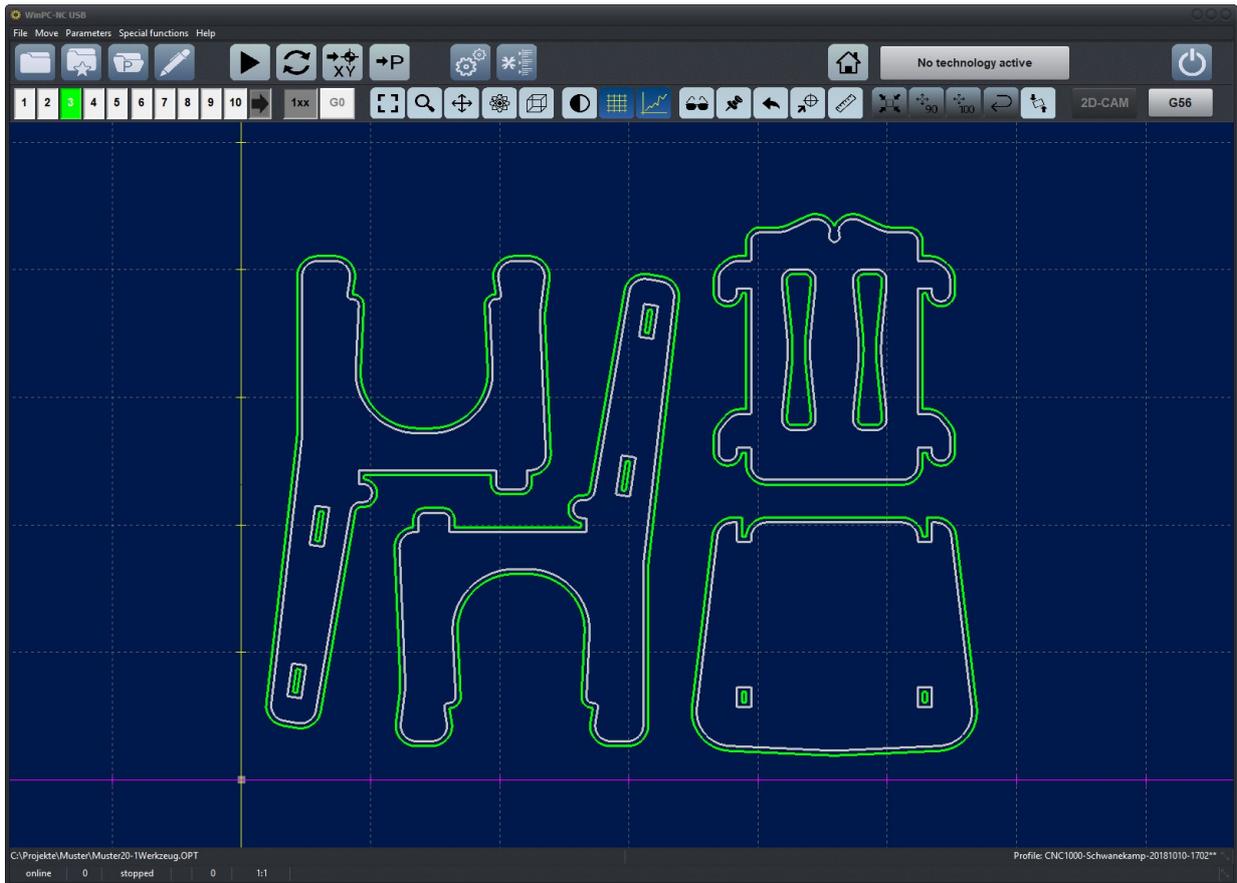
Calculated tool diameter offset

Approach/leave distance

This setting forces *WinPC-NC* to add a moving in line from outside to the contour and even to add a small leaving line at the end of the contour. This makes the edges of the contour perfect

Include original data to file

WinPC-NC can include the old and original vector data to the newly created file and shows it in gray color in graphics. This makes it very easy to control the newly calculated radius compensation.

Contour with move in/leave distance

Optimizing empty moves

Prior to the output of the calculated data in a new NC file, *WinPC-NC* tries to optimize (remove) any empty moves between lines and contours. Care is taken to ensure that any element is machined only once. This saves processing time and helps to decrease the milling time.

Tool data is made in the second window of CAM functions, e. g. radii for path correction and definition whether contour is placed inside or outside.
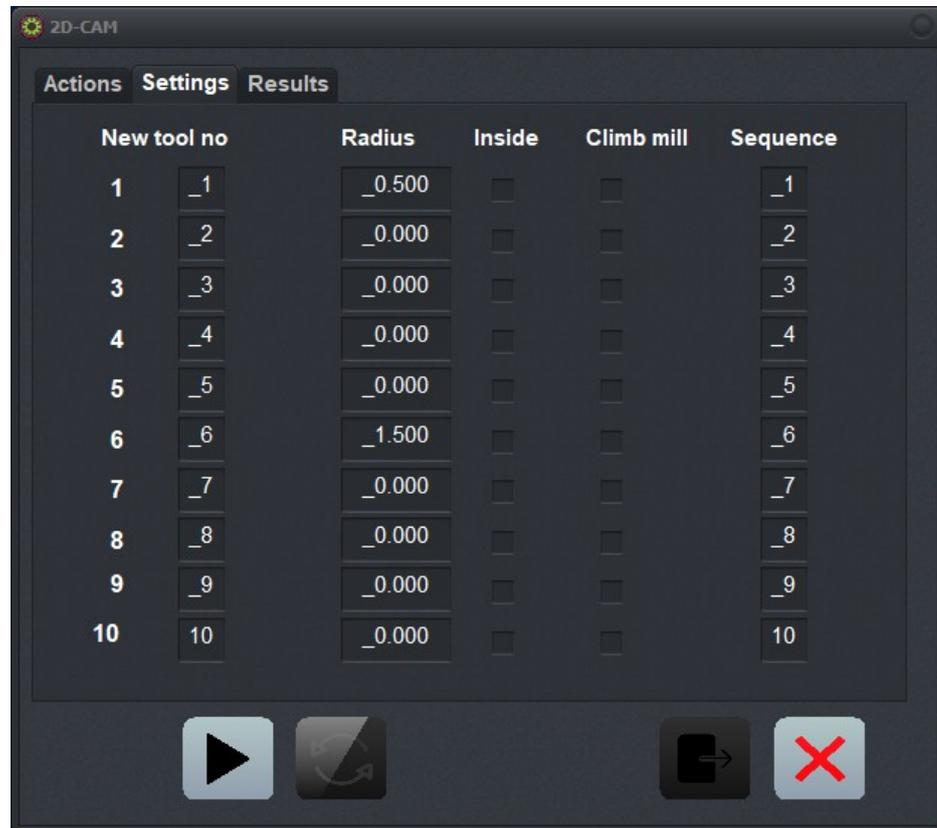
Setting options - tool data

New tool number

These input dialog boxes are responsible for resorting or summarizing existing tool or pen number settings prior to any other functions. It is often advisable to summarize several drawing components for a tool which is to be machined for more effective processing later on.

Radius

The radius of each tool is specified for the radius compensation. The radius is used correctly to handle contours from inside and from outside.

Inside

This button determines the direction of the new, relocated path. The line offset is made along the inside if you have clicked *Inside*, otherwise the line offset is made along the outside.

Climb milling

When this checkbox is check, climb milling will be used. Otherwise, conventional milling will be used.

Sequence of operations

The START button begins any (re)calculations. A progress bar will appear while the calculation process is running. The actual state of the process and the provisional results are displayed on the result sheet of the dialog box. Cancellation can always be effected by clicking to the corresponding button.

newly created file is loaded automatically

After **WinPC-NC** completes all of it's calculations and sorting, it creates a new 2D file in an internal format. The name of the file maintains the project name however the extension of the file will be *.OPT.

It is always possible to return the file to its original state (ABORT). For these procedures the buttons *START*, *RESET*, CANCEL and *DONE* are applicable.

Start   Reset   Done   Cancel

Having successfully calculated and generated the new file, the contours are immediately visible in the *Display Area*; the results can be checked. Additional changes and recalculations may be performed. Activate *Done* for accept the results.

Displaying the results

functions can be selected to individual needs

Manipulating the NC data can be performed in any order.

For example, in order to achieve a better surface result with mill-ings and engravings, you should clean up the data, join contours or lines, and remove duplicate operations.

4.3. Example 1

For demonstrating correct use and optimum employment of the most important functions, please take note of the illustration and step by step instructions.

We load a chair in *HPGL* format:
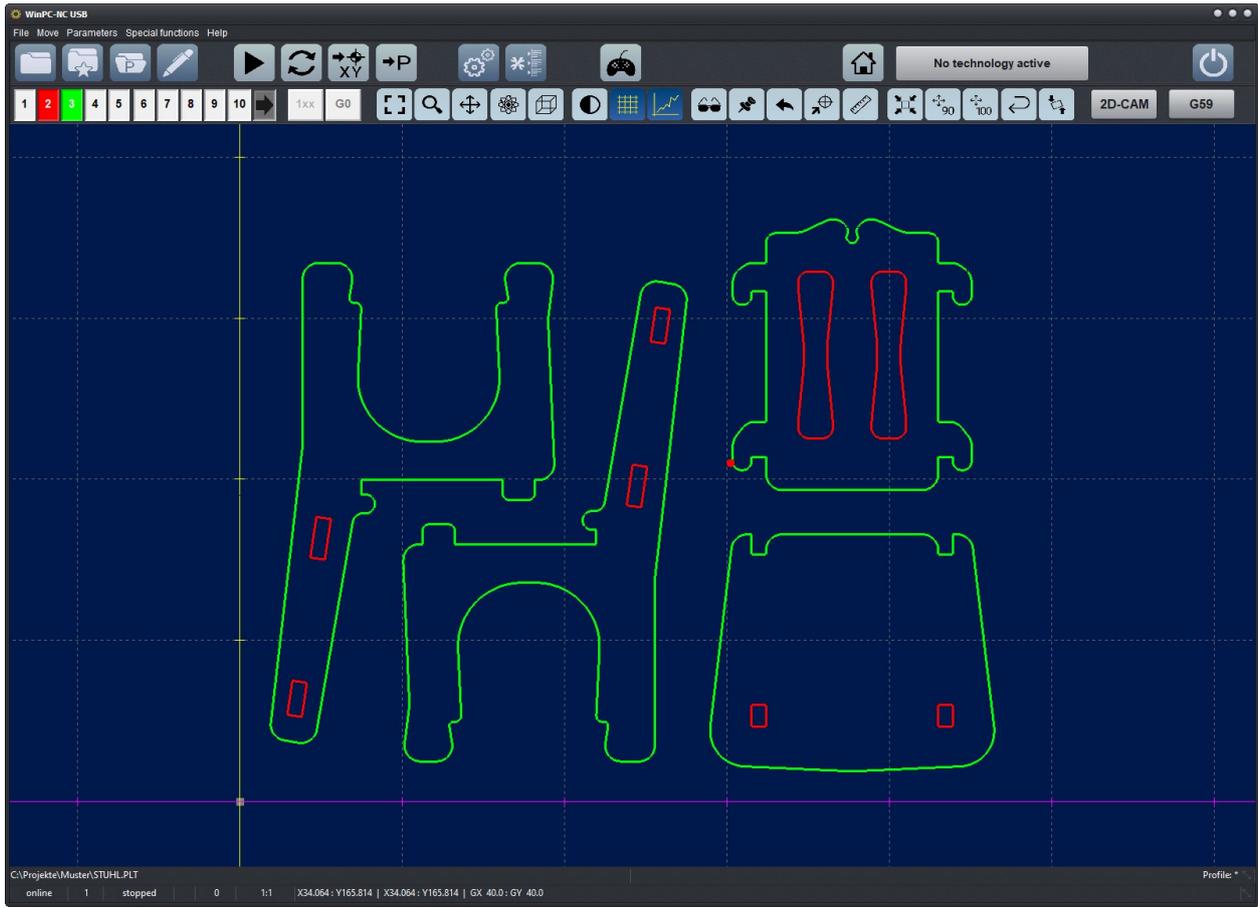
CAM-Functions : Loaded DXF- oder PLT-file with two tools

1. Two colors are present, indicating that the milling involves two tools. The interior contours are offset to the inside with a 1.6 mm tool; the outside contours will be milled with a 2mm tool. Settings are as follows:
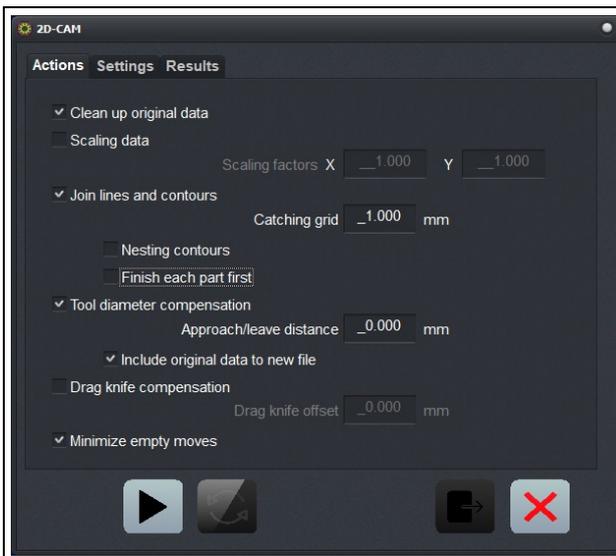
2. We define the values for tools 2+3, and start a search for contours, calculate radius compensation, and optimize any empty moves. The result is…
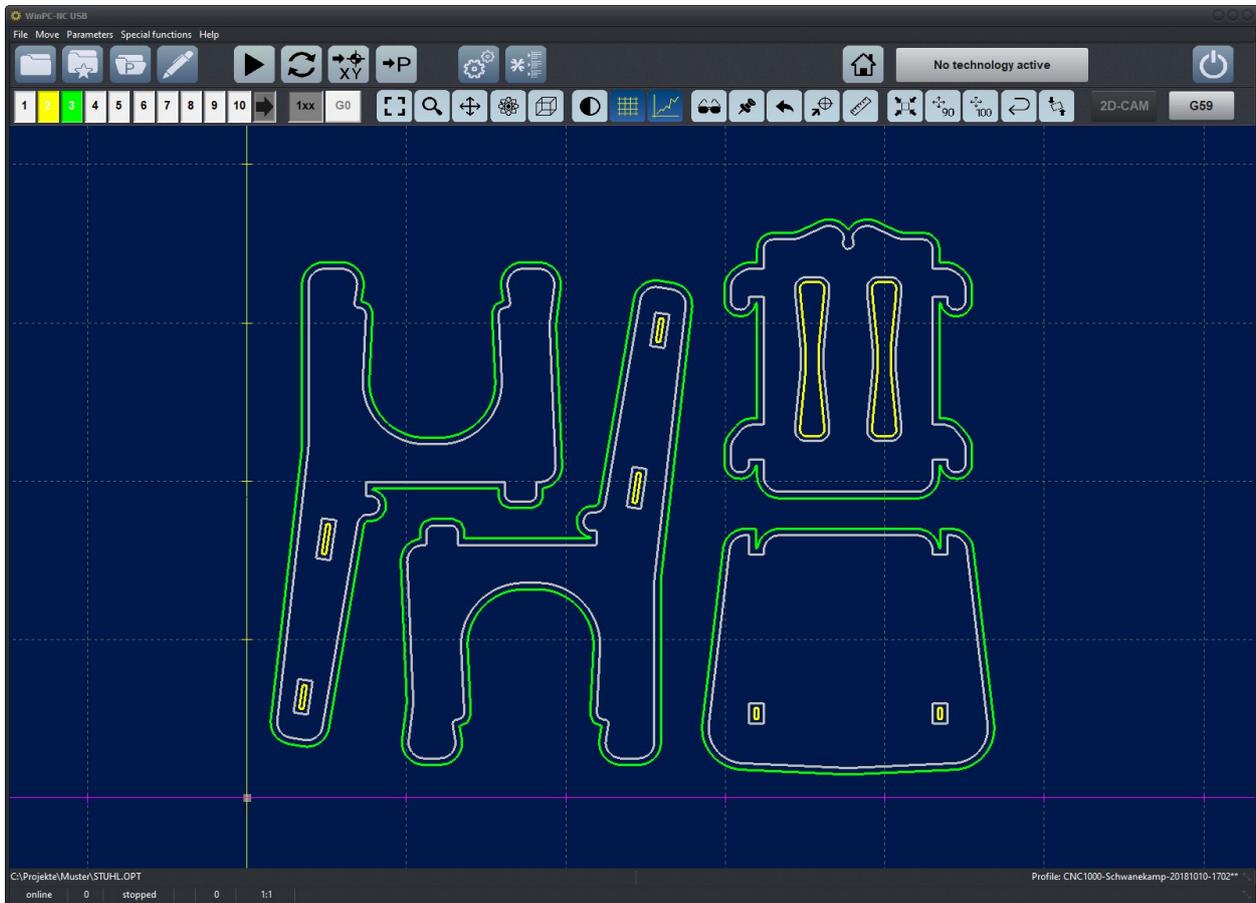
CAM-Functions : Calculated contours

3. If we want to mill the job using only one tool - no tool change - calculations have to be redone. The function is reset, tool 2 is disabled, tool 3 is defined, calculations are redone, including the radius compensation.
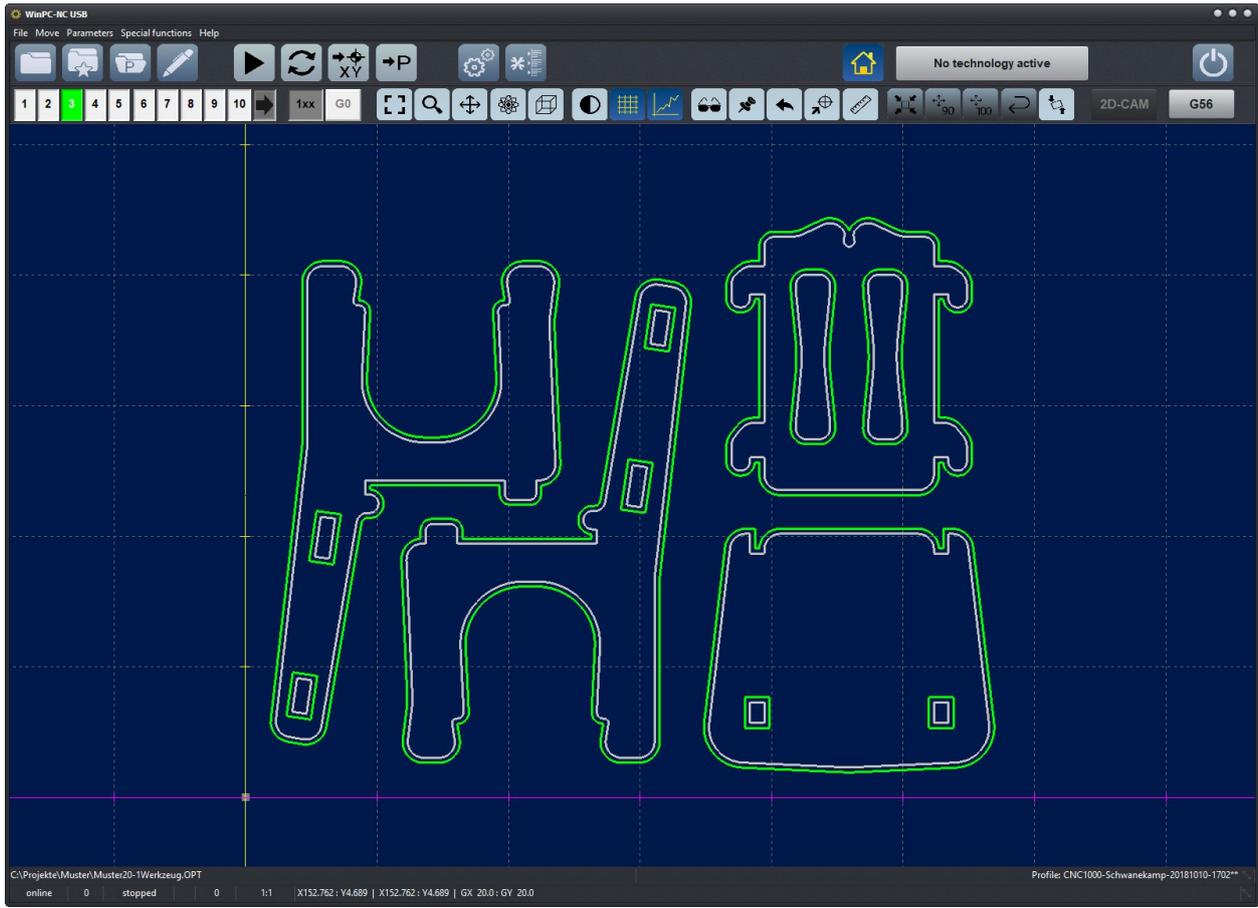
CAM-Functions : Calculated contours for only one tool

4. Notice that the interior contours are improperly milled on the outside. We still activate the button for the automatic search of surrounded contours and start calculation for the last time.

5. Now, the result fulfills our requirements and expectations, sequences are correct, and all work pieces can be machined with our 2mm tool.
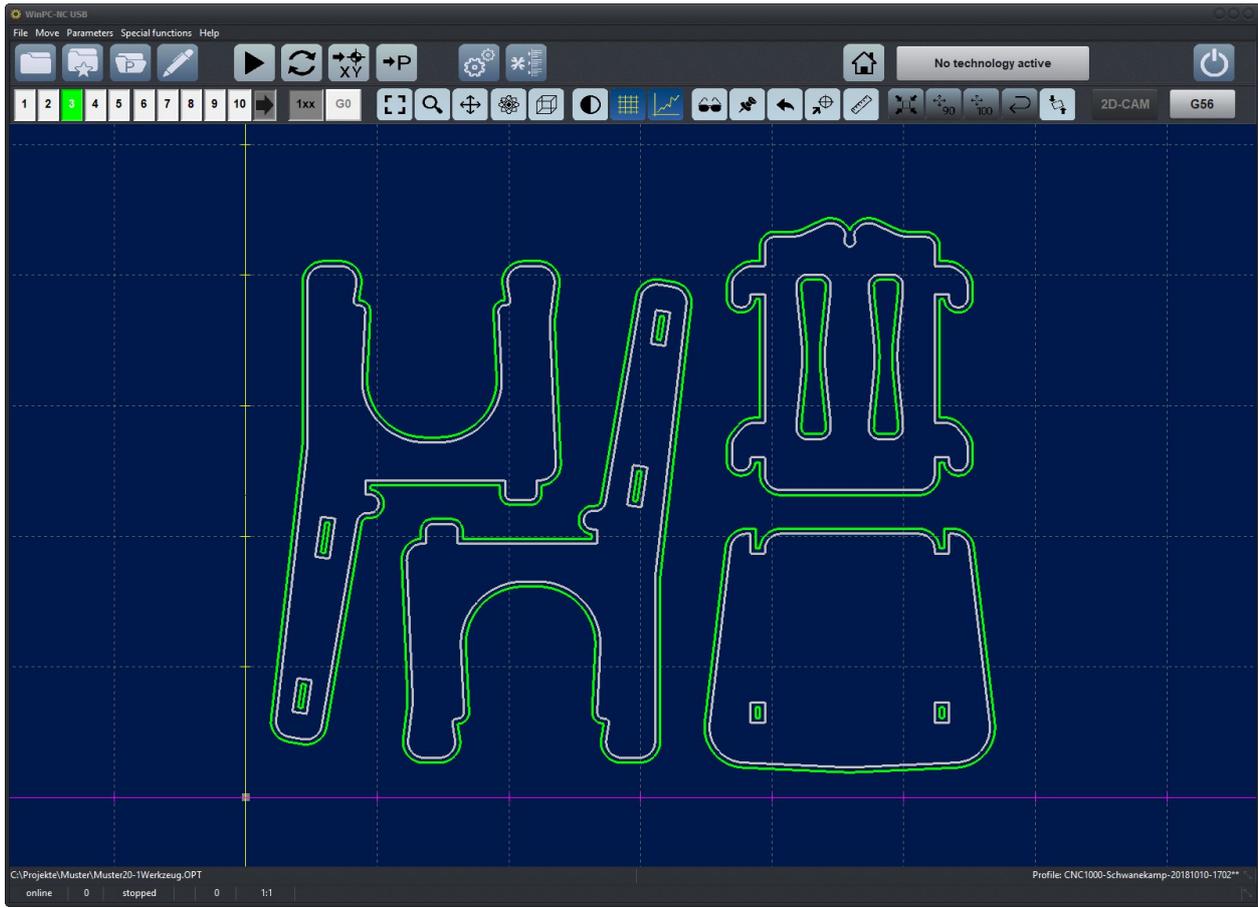
CAM-Functions : Calculated contours for only one tool with the correct inside/outside assignement

4.4. Example 2

1. We are loading a complex DXF-file and intend to mill all contours with a 1mm tool and place the contours independently.

CAM-Functions : Loaded DXF-file with larger quantity and many contours

2. Settings in both dialogues are as follows:
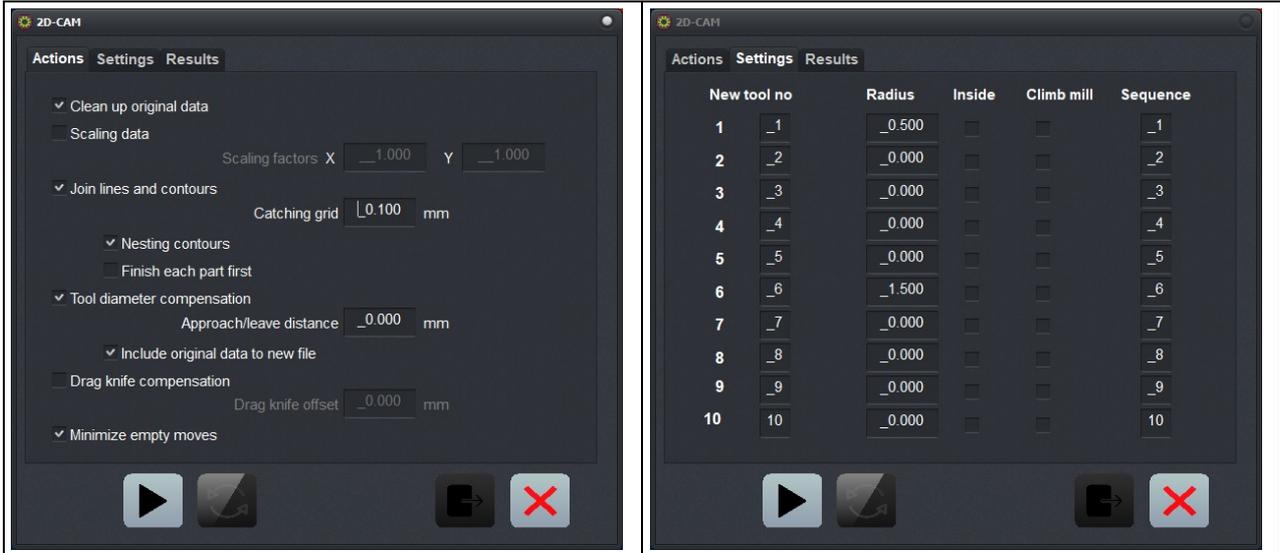
3.   At first, the result seems to be perfect. The surrounded contours as well as the offset paths to the inside and outside are clearly displayed. However, after zooming, some contours have not been correctly recognized and calculated.

CAM-Functions : Small gap in the contour

4.   There is a spacing between two successive lines and it is large than the set catching grid.

5.   We extend the catching grid up to 0.2mm and recalculate. A perfect result is achieved and milling can begin.
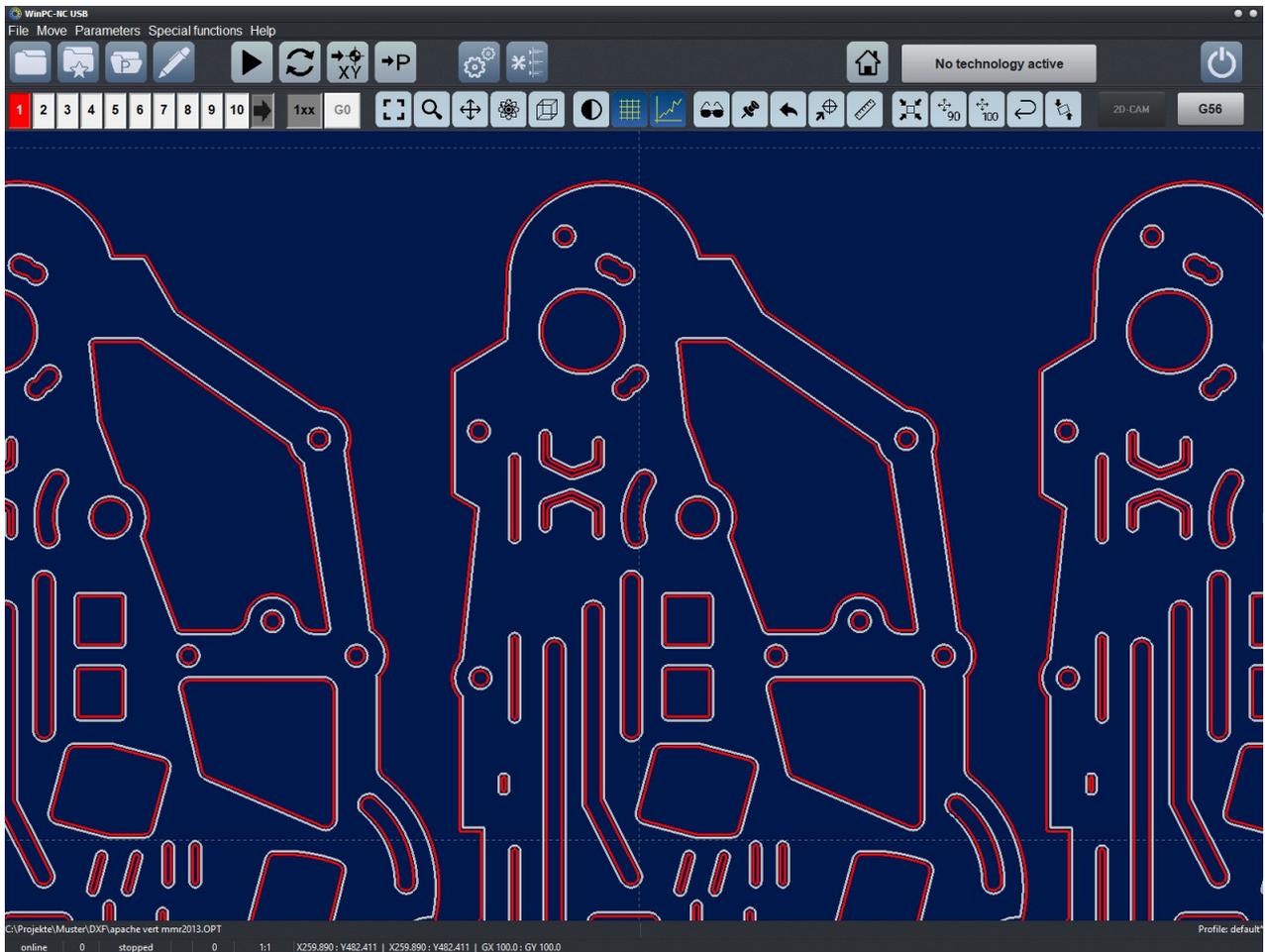
CAM-Functions : Perfect calculation over all contours

5. Parameter Settings

5.1. General

The parameter structure is sub-divided into two levels. The main levels comprise pages as coordinates, tools and technologies and describe the higher-level group.
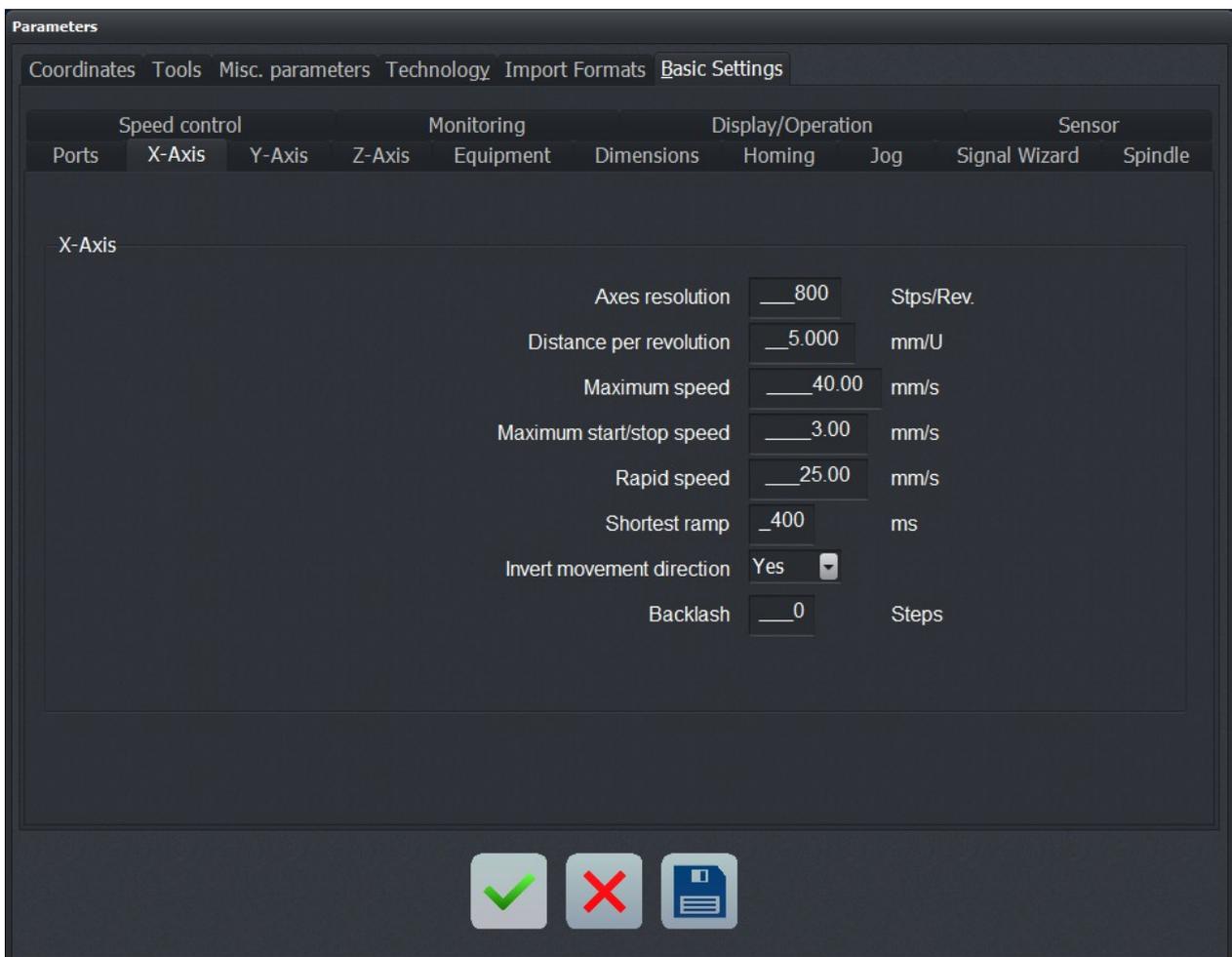
Parameter-Basic settings

Some groups are divided in additional sections and display additional windows concerning settings which are presented in subcategories.

> **If the software is delivered bundled with a machine, in most cases the basic settings are properly pre-defined and chapter 5.2 can be skipped.**

5.2. Basic settings

Adjustment of mechanical and electrical characteristics

Basic settings are those parameters that are not expected to change very often (e.g. language choice, mechanical and electrical characteristics, user interface choices).

All basic settings are grouped in this category and as a rule they remain unchanged with later use of the machine, exceptions may be made.

Normally, the page for basic settings is hidden. However, it can be displayed by a click to basic settings in the menu in the category parameters .

5.2.1. Basic settings - Ports

Except for definition of the connected joystick, keypads or manual wheel in **WinPC-NC** does not require any additional data concerning interfaces. The used USB port is automatically determined and communication to **ncUSB** is made.

Depending on the used version, **WinPC-NC USB** or **WinPC-NC Professional**, different setting options are displayed or parameters which are not relevant are hidden.
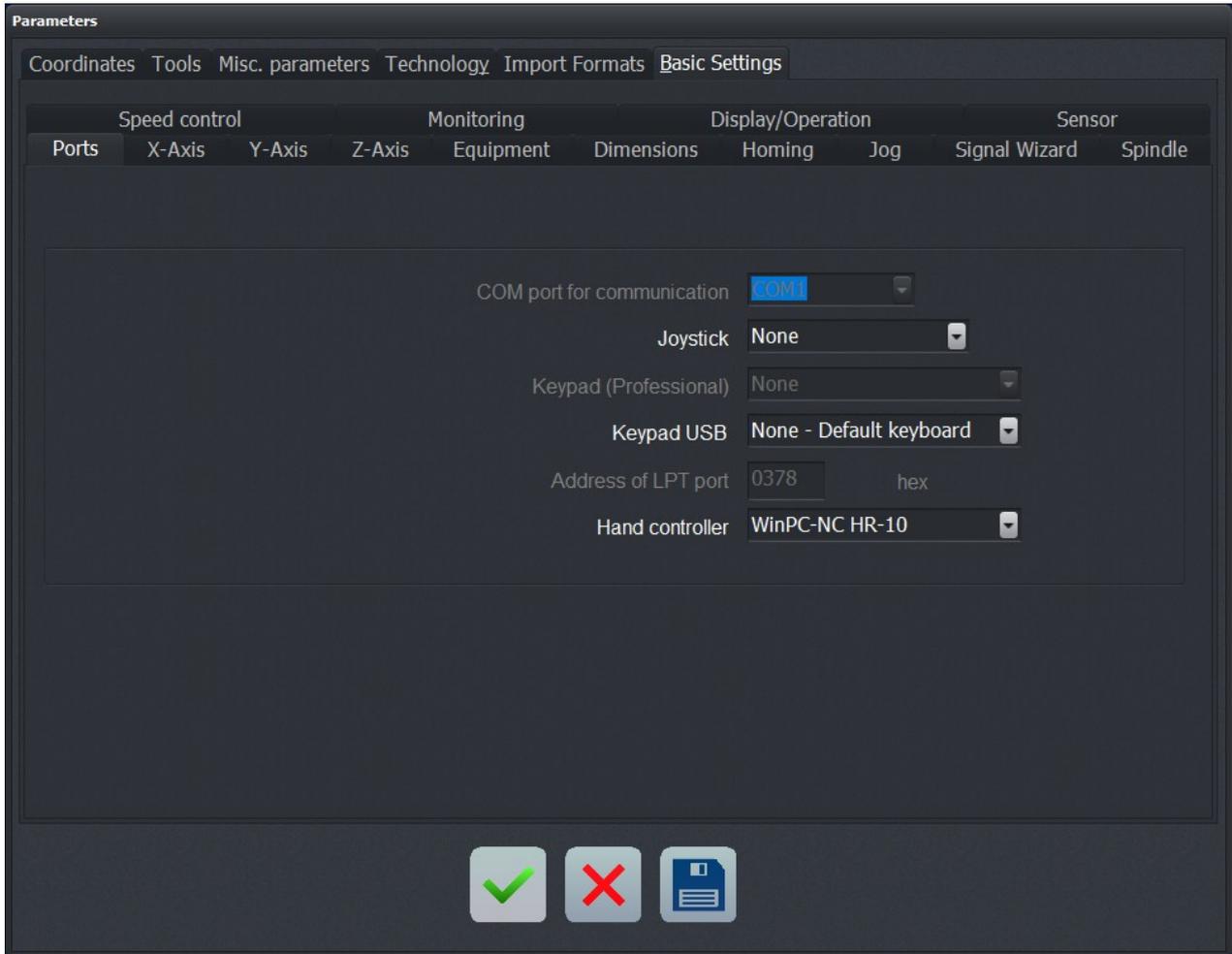
Parameter basic settings - ports

COM port for communication

At *WinPC-NC Professional* this parameter defines the serial input for connecting a nul modem cable with the control or the axis controller CNC-Control. You can also use a regular USB-RS232 adapter if there is no original COM port on your computer.

WinPC-NC displays in the status bar the message offline if the port has been incorrectly adjusted, if the wrong cable has been used or if it is defect or if control is not ready for operation.

Joystick

If you want to use a joystick or 3D Space Mouse with your machine, specify the connection via the pop-up menu.

The joystick or 3D Space Mouse must have its drivers loaded prior to choosing it here. It must be calibrated via SPECIAL FUNCTIONS-CALIBRATE JOYSTICK before use.

Keypad (Professional)

WinPC-NC Professional can easily be used by an external keypad. This keypad is a compact gadget and can be directly connected to the axes controller by a cable of up to 5 m length.

Keypad (USB)

A small, numeric keypad (including generic USB keypads) can be used as a remote control for *WinPC-NC*. Make the selection appropriate to your keypad.

Pendant (Hand Controller)

WinPC-NC supports only those pendants listed in the pop-up. The *HR-10* pictured is available directly from *WinPC-NC*.

Pendant hand wheel *HR-10*

5.2.2. Basic settings - Display/Operation
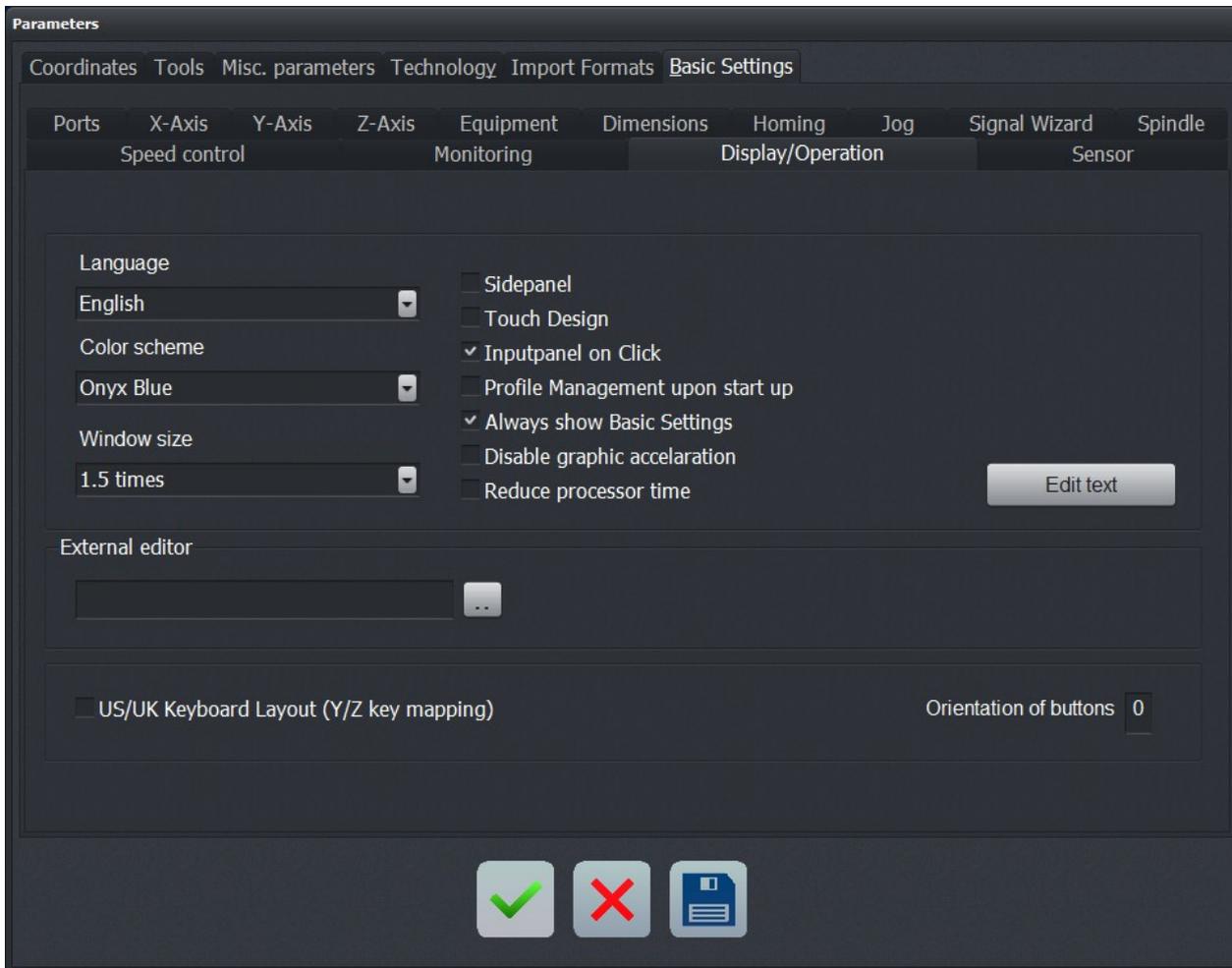
Parameter Basic settings - Display / Operation

Language

WinPC-NC is multilingual. The standard version already includes a few languages, and additional languages can easily be bolted on if required. The available languages are listed in a menu.

Due to an integrated text editor in **WinPC-NC** it is easy to add additional languages and thus individual text translations can be easily realized. Please contact us for detailed information concerning the language editor.

According to March 2021 the following languages are available:

german	czech	chinese (taiwan)
english	macedonian	chinese (traditional)
french	slovenian	turkish
spanish	italian	bosnian
polish	portuguese	russian
hungarian	norwegian	corean
greek	croatian	
dutch	serbian	

A language selection takes effect immediately. Click SAVE to make the selection permanent. Please be aware that some messages are part of Windows and may not be properly localized for the language selection.

Color scheme
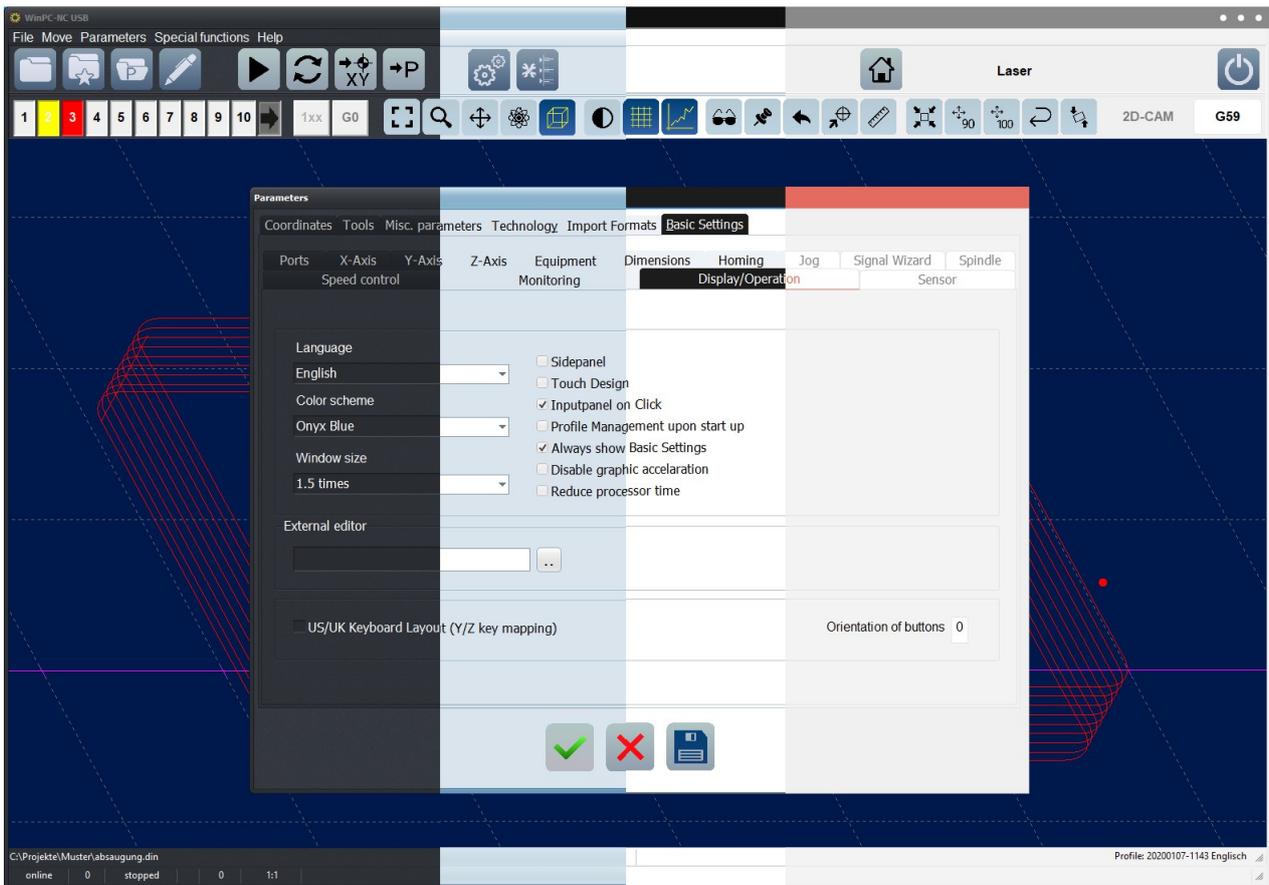
Four different *WinPC-NC* color schemes

Choose the color scheme that best suits you. Please contact us concerning other possible choices*.* The illustrations in the manual are set with *Onyx Blue* style.

Windows size

Font sizes and windows sizes in **WinPC-NC** can easily be adapted as required by this parameter. Independently the main screen can be extended or minimized by using the mouse in the usual manner.

Changing this value means that **WinPC-NC** requires a reboot.

External editor

WinPC-NC includes a high-performance editor suitable for editing NC files. If you have a favorite or preferred editor, select your editor here and **WinPC-NC** will invoke it anytime an editor is required.

US keyboard layout (switch YZ keys)

Jogging in the direction of an axis (i.e. X, Y, Z) can be performed by hitting the letter key associated with the desired axis. Hitting CTRL plus axis letter increases the jog rate.

European and North American keyboards switch the position of Y and Z keys and this may make it difficult to use the keyboard for jogging. Selecting this checkbox corrects for the switched layout.

Button orientation

adjust moving directions to arrow keys

The button orientation parameter rotates the buttons in the jog button palette by a multiple of 90 degrees. Adjust the jog button palette orientation so that it matches your machine.
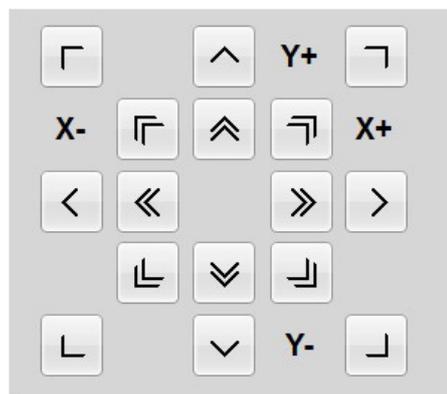
Button orientation 0   Button orientation 1
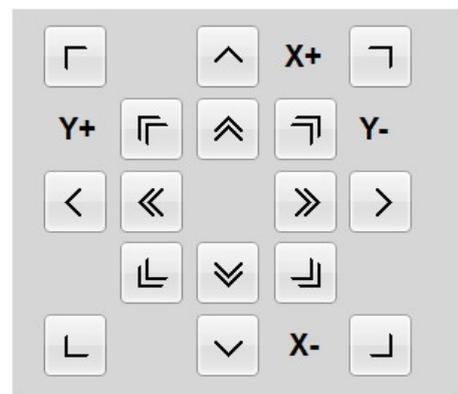
X-Y buttons are rotated by 90° each by position 2 and 3.

Side panel in main display

Only in the standard view of *WinPC-NC* is the so-called side pa-nel open in the main screen after activating this parameter. This means that either all functions of Jog move are directly presented or during a job the related control and display panels.
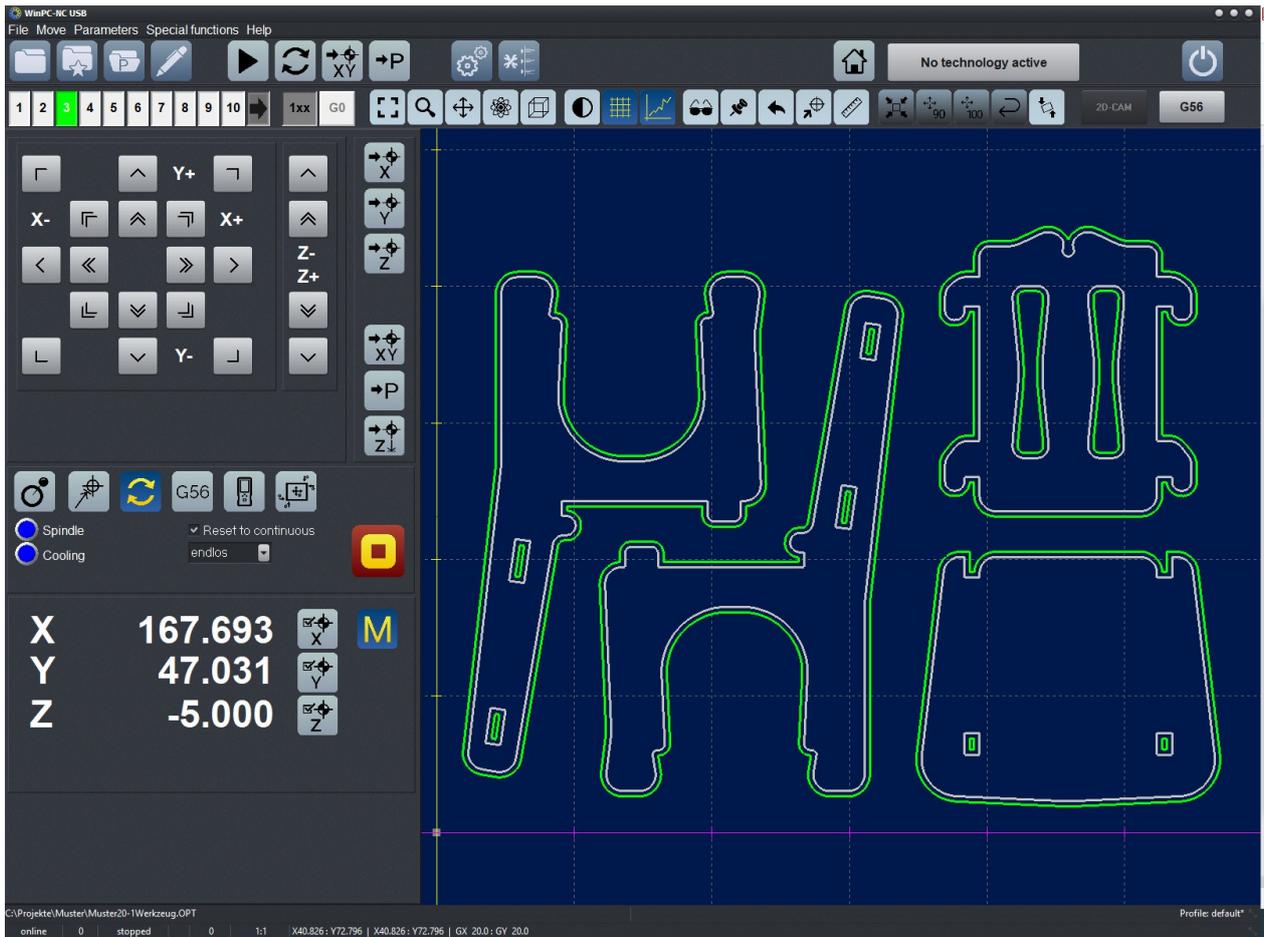
WinPC-NC with sidepanel - Jog
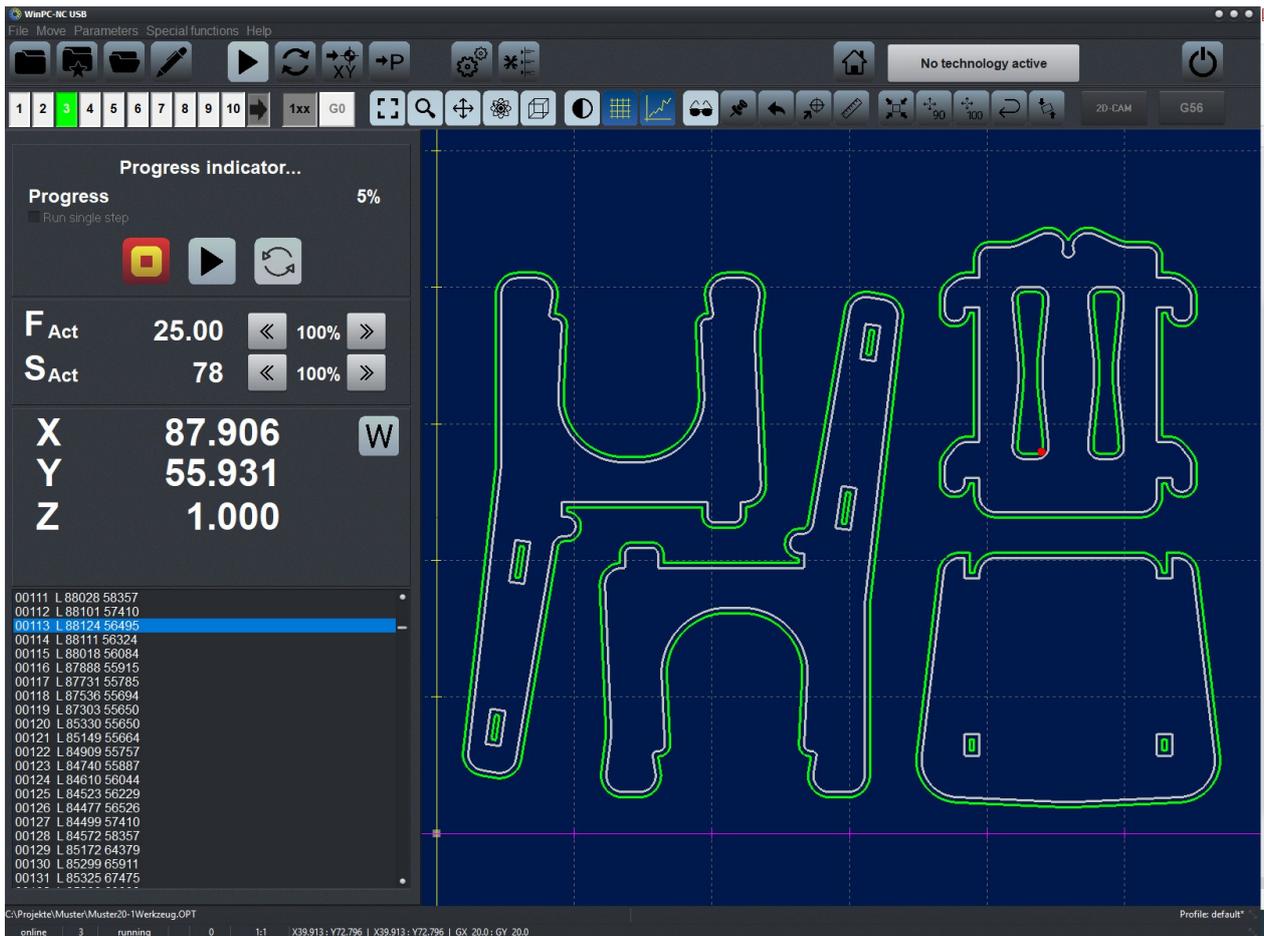
WinPC-NC with sidepanel - job view

Touch optimised display

WinPC-NC provides a view which has been optimized for using a
touch panel. After activating this parameter and restarting
WinPC-NC this view is displayed.
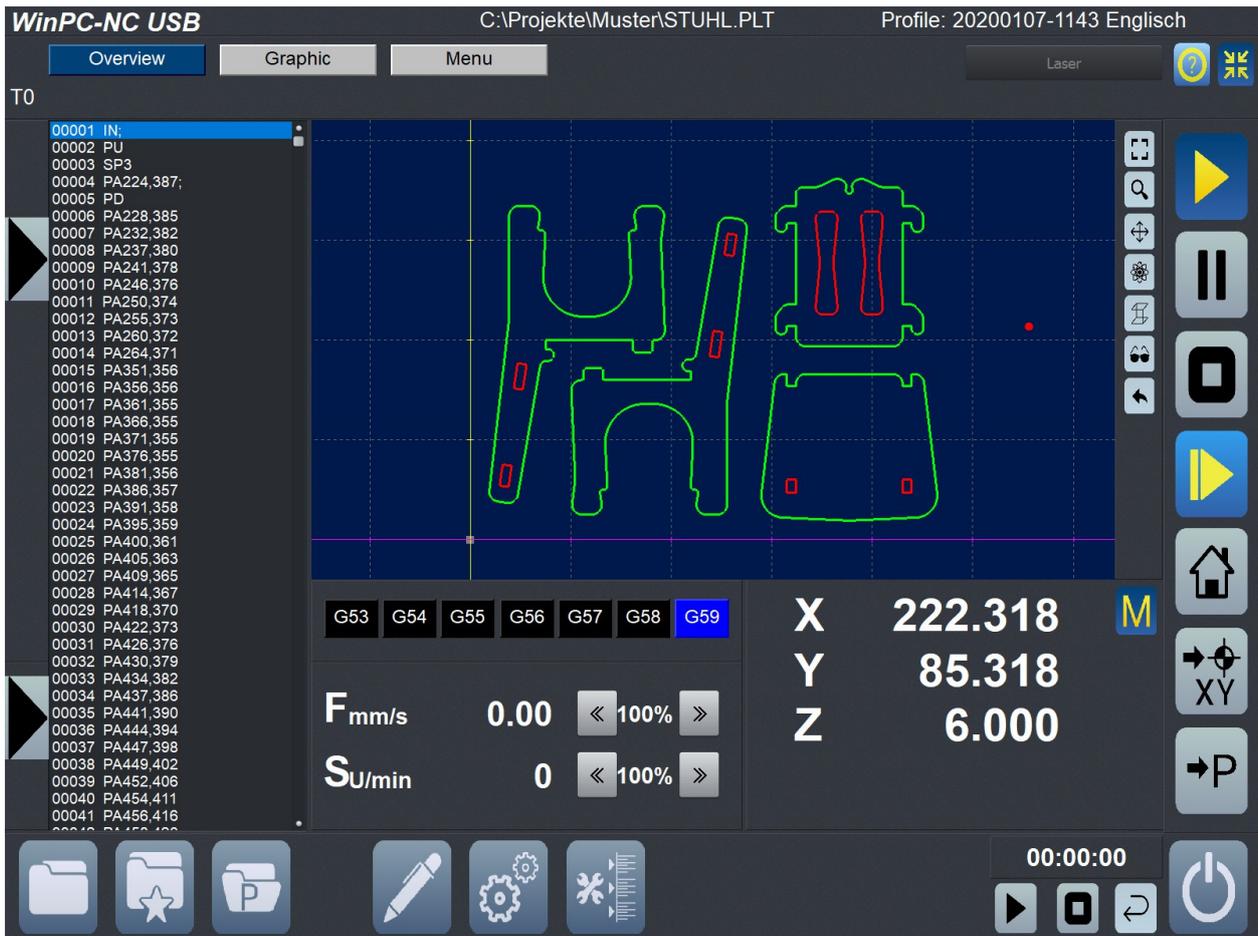
WinPC-NC Touch display - main screen

Input panel with click
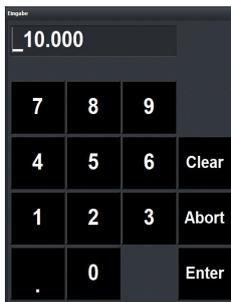
Entries can also be made by a panel which is usually opened by a double-click. Using a touch panel, we recommend opening the input panel by one click.

This is activated by this parameter. After restarting ***WinPC-NC*** all functions are available.

Disable graphic accelaration

Most graphic boards can operate with so-called graphics acceleration. In case of incorrect graphical display, it is advisable to deactivate this function.

WARINING !

Especially some graphic boards of ATI may cause incorrect displays and compatibility problems. If this is the case, there will be no graphical display after a file has been loaded and at the end of the program an error message will be displayed with "Thread error".

The problem should be managed by deactivating the graphics acceleration

Reduce computing time

It is not advisible to use computers which do not meet the latest technology standard, as they are often unable to process complex graphics. To manage this problem, we recommend the activation of this parameter. This means that graphics are not updated in the standard operation mode during the movement of axes (e. g. red dot for current position). If axes are stopped and do not move, graphics are refreshed in longer intervals.

Always show basic settings

The basic settings will appear in the normal parameter dialog if this checkbox is checked. Sometimes it may be a good idea to hide machine specific settings to prevent accidental modifications.

Profile Management upon start up

WinPC-NC will load and use the specified profile each time the program begins.

5.2.3.   Basic settings - X,Y, Z axis
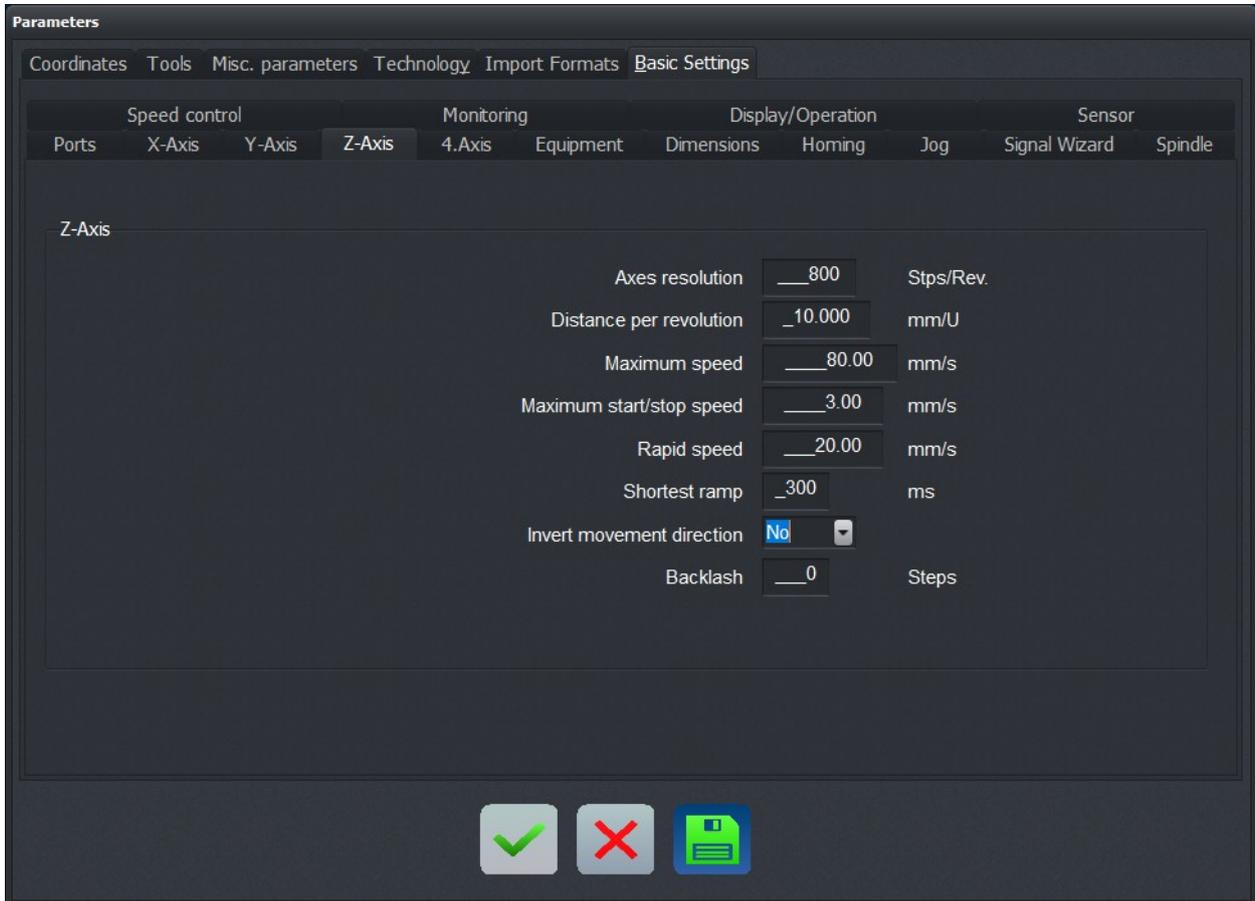
Parameter Basic setting Z-axis

Axis resolution

Specifies the number of motor steps per revolution. Any gearing or step up/down ratios should be calculated with this number.

Distance per revolution

Specifies the distance travelled each motor revolution. Units are millimeters.

Maximum speed

Use the MOTOR TEST function to ascertain the maximum speed of each axis. This represents the absolute top speed with which an axis can be moved. The units are millimeters per second.

Maximum start/stop speed

Specifies the fastest speed (in millimeters per second) with which the stepper motor can move. This value is used to maintain the highest possible machine throughput. Sharp edges or corners do not have to bring the machine to a standstill; movement can continue at the start/stop speed.
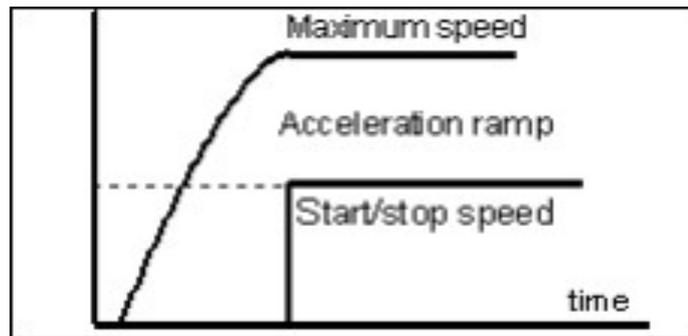
Start/stop-speed and ramp

This speed **must** be obtained by experiment via the MOTOR TEST.

Switch off the ramp and run acceleration tests until step losses occur. For safety reasons, enter 30% of the measured rate.

Values for maximum speed and start/stop speed have a significant impact on the look-ahead contouring control, which analyzes the following movement by complex Look-ahead functions and always tries to compute the optimum speed.

Rapid speed

speed at unproductive movements

The rapid speed specifies the fastest usable speed when moving from one job location to another – and the tool is not in use.

The rapid speed will be used whenever possible.

Shortest ramp

Specifies, in milliseconds, the time it takes to go from standstill to maximum speed (or maximum speed to standstill).

ramp length define in milliseconds

Each motion is broken into three phases – ramp up, maximum speed, and ramp down.

Using ramping prevents step losses and safely maximizes the highest movement speed.

Each motion begins with the speed being increased until the maximum speed is reached. The maximum speed is sustained for the longest time possible then the speed is reduced until the motion comes to an end.

The slope of the ramp is constant for all movements - the ramp time is shorter for movements at slower speeds.
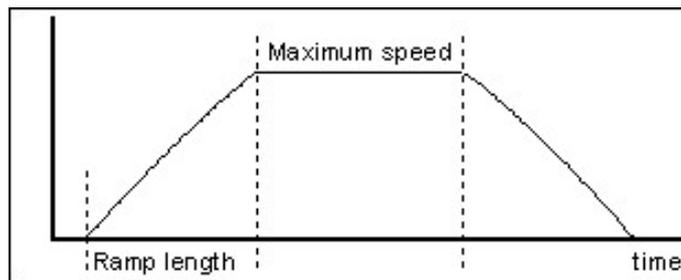
Speed profile of a movement

Invert movement direction

If a motor is moving in the opposite direction than expected, there are two solutions possible:

Invert movement direction

- Swap two of the motor wireing connections

- Check this checkbox - the motor direction signal will be inverted.

Backlash compensation

In some systems, reversing direction results is a positioning error. While these errors are usually small, they can add up over time and lead to unacceptable results. This parameter can be used to compensate for this issue (backlash compensation). The units are additionally indicated steps.

For all axes parameters are available for defining open circuit steps. The number of these motor steps are additionally indicated for each reversal.

If you are unsure as to use this parameter, or have a closed loop system, leave this parameter set to zero.

WARNING !

Backlash can not reliably be determined with testing. It must be always remeasured by a dial gauge. Although *WinPC-NC* is able to compensate defined bachlash (second best solution), it is recommended to set the axis, if possible, free from backlash.

5.2.4. Basic settings - 4.-axis
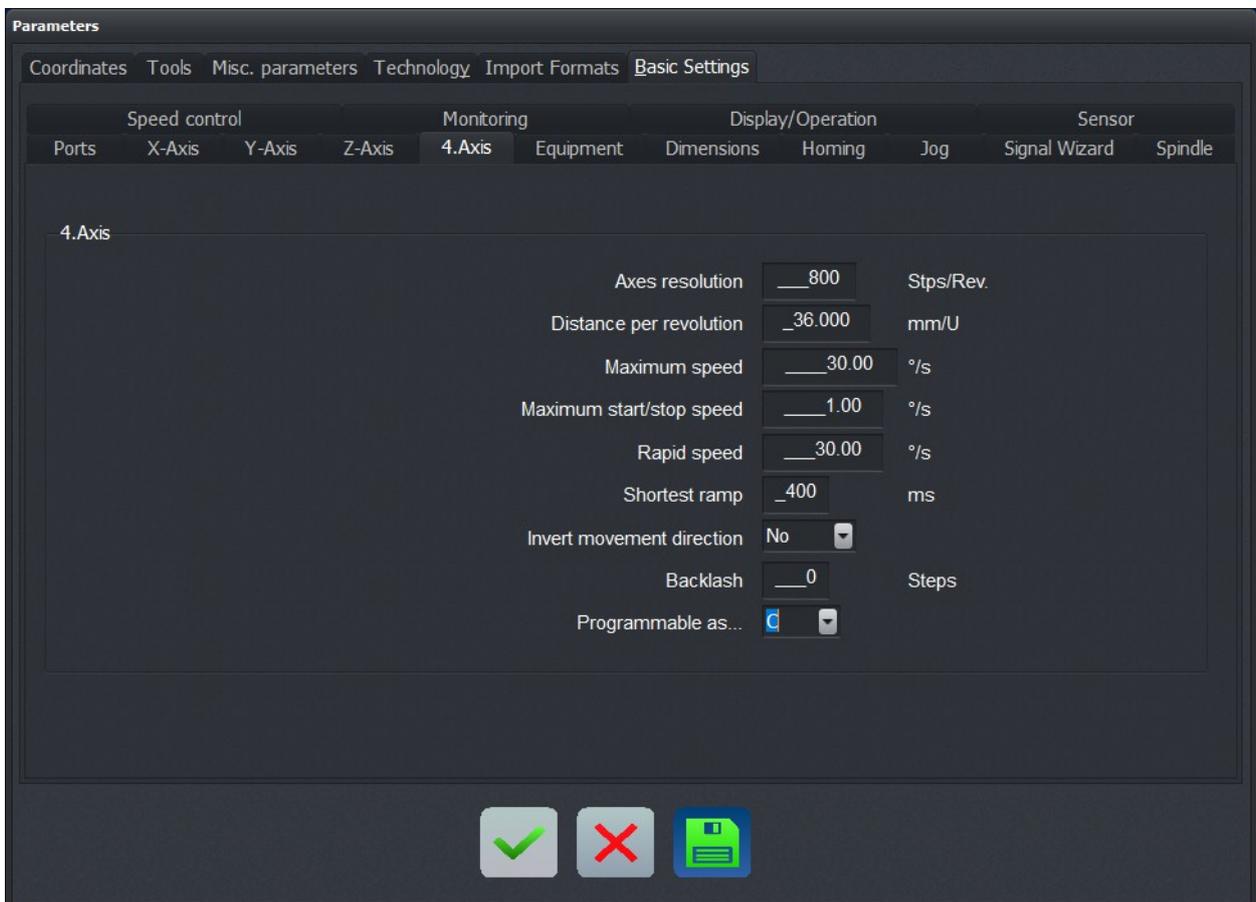
Parameter Basic Settings 4.-axis

WinPC-NC is able to manage a 4th axis. Programming maybe implemented by a G-code program. Another possibility is the automatic direction rotation of the tangential axis by *WinPC-NC*.

Prior to using the 4th axis it has to be enabled. Otherwise the parameter settings are not available.

Some of the parameters, such as speeds, inverting the direction and reference switch are acting in the same way as for axes XYZ and are not explained here.

Axis resolution/distance per rotation

The resolution parameters operate in the same way as the standard X, Y, Z axes. If you enter 360 or 36 as the distance per rotation, then you can program the 4th axis as a rotational axis in degrees or tenths of a degree.

Programmable as...

Letter for programming

The 4th axis is addressed in G-code programs via a letter.

Axes parallel to X, Y, and Z are designated, respectively U, V, and W. Rotations around the X, Y, and Z axes are designated, respectively, A, B and C.

3D applications often use E for the filament feed.

Parameter settings and programming of parallel axis with letters U, V and W are made in mm and mm/sec. Settings of rotary axis with letters A, B and C are made in degree and degree/sec.

5.2.5.   Basic settings - Equipment
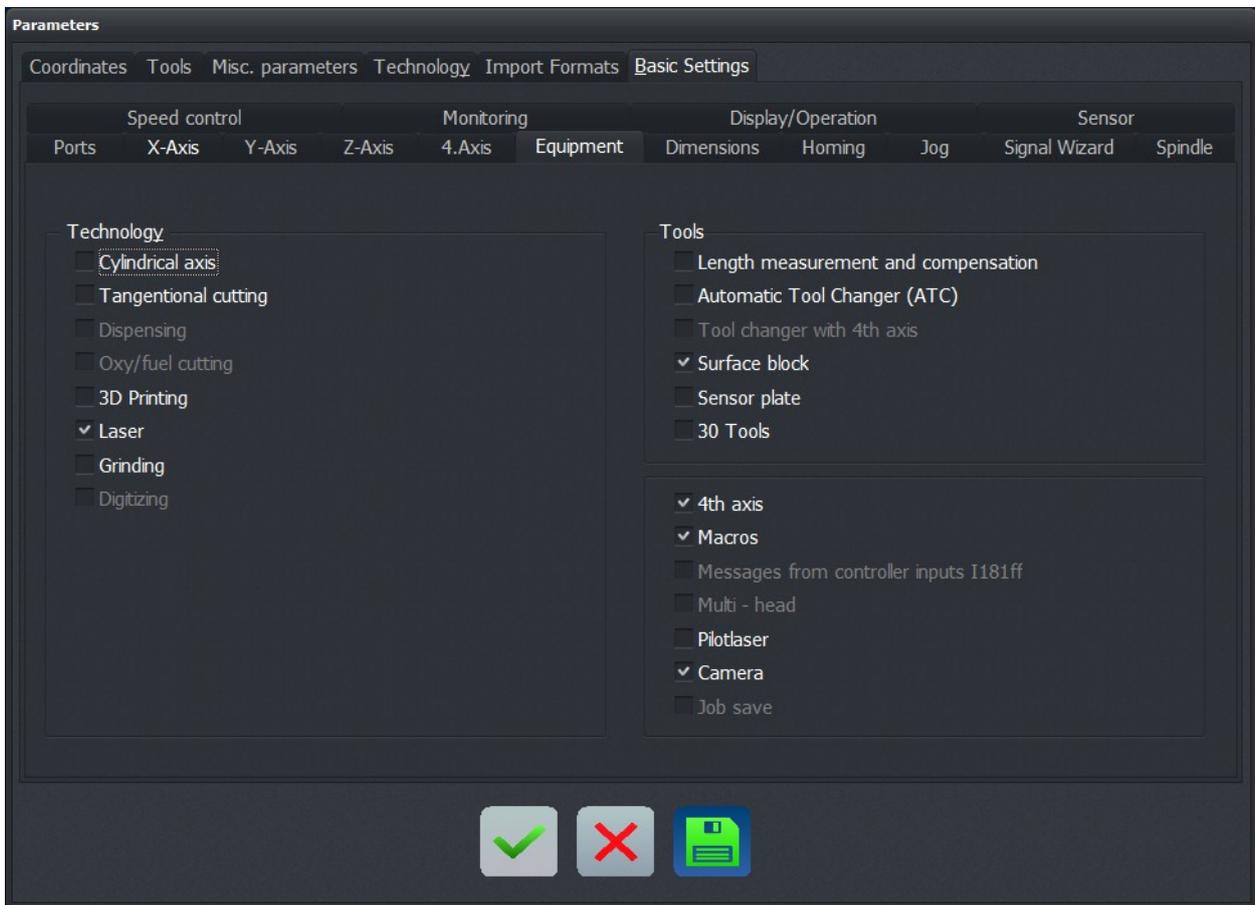
Parameter-Basic Settings-Equipment

On the page for equipment all auxiliary components can be selected, which are included in delivery and which you intend to use. For example, you can select a 4th axis as well as other multiple functions.

Depending on the selection you have made, additional parameters and settings are available and displayed. Some of the selected components are displayed later on in the technology page and can be individually activated as required.

Thus the machine can be easily reset for specific jobs and the relevant technology can be selected by defining detailed settings, e. g. printing in 3D or laser engraving.

Greyed out technologies are available in *WinPC-NC Professional* only.

The following equipment settings and additional functions can only be used with *WinPC-NC Professional*:

- Dosing/dispensing technology

- Flame cutting technology and Plasma cutting technology

- Monitoring of inputs and freely definable message displays

- Tool changer with 4th axis, e. g. as drawer or turret magazine

- Digitizing of non planar surfaces for reproduction

- Double and multiple heads with several Z-axes and automatic switch-over

Technology

The equipment settings present those *WinPC-NC* technologies that your machine *may* support. When a technology is enabled, the parameters and options specific to it will be visible and can be adjusted.

The specific configuration of your machine may limit the *WinPC-NC* technologies available to you. Some machines may only support a single technology (e.g. milling); other may be configurable for multiple technologies (e.g. 3D Printing, Laser). Check with your machine vendor for the specific details of your machine.

Enable those technologies that are applicable as you need them.

Tools

The tools section of the equipment settings presents the check-boxes for enabling automatic length measurement and length compensation, Automatic Tool Changer (ATC) support, and sensor block support (assistance with setting Z zero at the top of a work piece).

If there are more than the default 10 tools you can work with up to 30 tools by activating the checkbox „30 tools".

The zero point of your workpiece can be determined by a touch plate. Additionally it is possible to execute a phase angle correction, if the workpiece is not exactly aligned to the axes (Warning: only active if input for probe has been defined). By activated parameter the tab "Probe" is added to basic settings and the parameter for the touch plate is released.

The function sensor block is made for automatic and contact free definition of the Z zero points (Warning: Only active if an input is defined for probe). By activated parameter the tab "Probe" is added to basic settings and the parameter for the sensor block is released.

Other equipment

If function *4th axis active* has been enabled, all functions in **WINPC-NC** which require a 4th axis are set ON/OFF.

Furthermore it is possible to use macros in order to execute easily self-defined sequences and moves in various programs.

For a more precise orientation in jog move you can use a pilot laser or a USB camera.

5.2.6. Basic settings - Dimensions
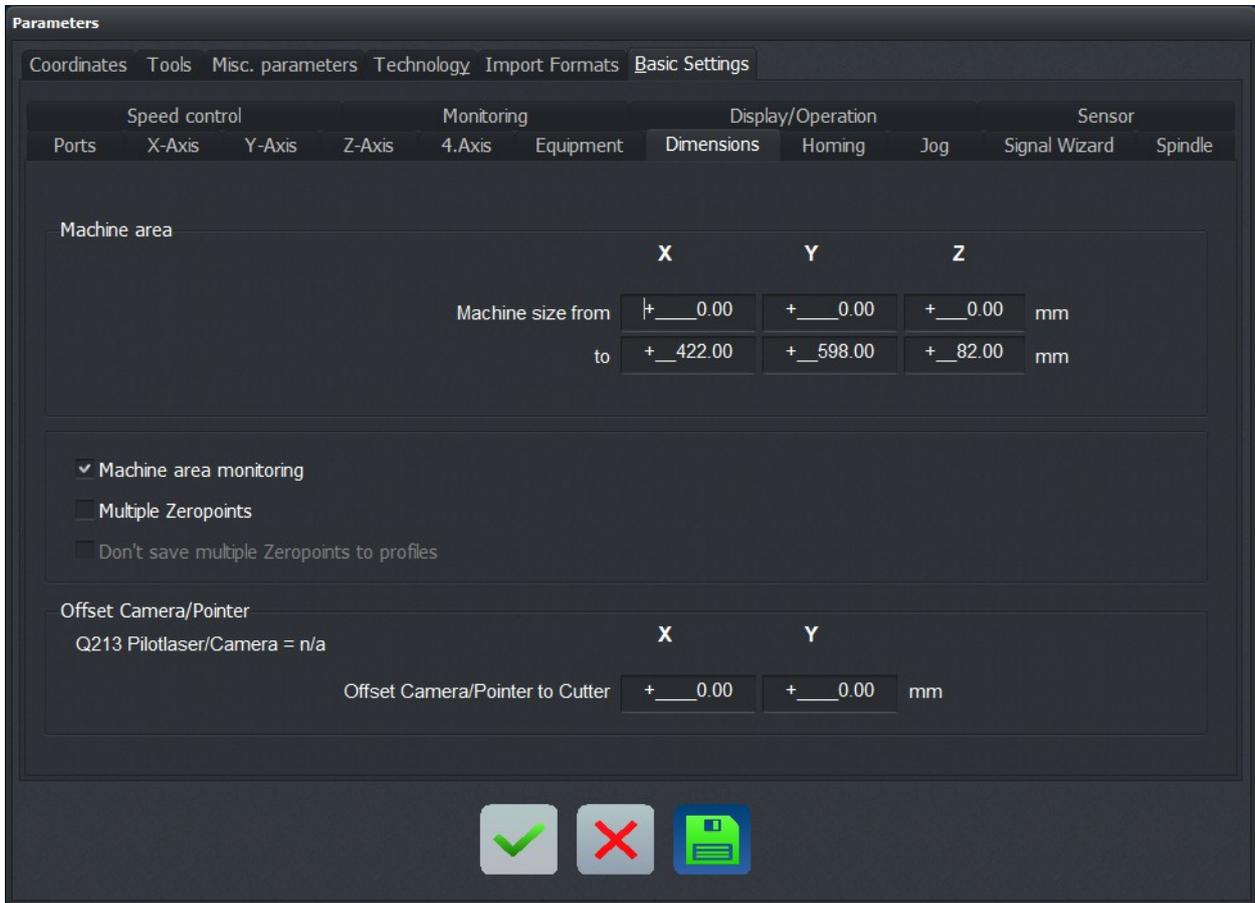
Parameter-Basic settings-Size/Dimension

Machine size and machine area monitoring

Machine size Specify the effective movement area of the machine. If machine area monitoring is checked **WinPC-NC** will monitor all movement, including manual jogging, against the machine size

Multiple zeropoints

WinPC-NC allows operation with multiple zero points (G54-G59). By activated parameter the required buttons, dialogs, etc. are enabled in the program.

Detailed information on how to use multiple zero points can be obtained in our homepage www.Lewetz.de in the menu *Help - HowTo – instructions* .

Don't save multiple zeropoints to profile

By activated parameter the coordinates of the different zero points are not saved with the profile function.

Camera usage

If a small camera has been connected and configured in Windows, the corresponding camera image can be used for exact determination of approached positions and it is easy to set the display of the camera on/off in the dialog of jog move. It is also possible to define a possible offset or misalignment of the camera axis to the tool axis.

*WinPC-N*C main window and jog mode with camera display

5.2.7. Basic settings - Homing

Parameter-Basic Settings-Homing

Home switch at ... end

Specifies which end of the machine the home switches are located. Home switches are most commonly installed on the negative end of an axis, however it is not unheard of to place them on the positive end.

Home point is…

Specifies the home position. This overrides the default where the machine initialization procedure determines the zero position of each axis by searching for the home switches.

It is only useful to set a specific reference position instead of a zero point if the reference switch is not at the negative end of the axis or limiting the traverse path but is set either at the positive end of the axis or moved to the middle.

Thus the axis zero point is moved to the corresponding end of the traverse path and positions can be easily programmed. And besides, it is possible to use positive coordinates.

Home offset

Once the home position of each axis has been determined (see the search process description below), it is best of add a small offset to the physical home position to ensure that the home switch is not tripped by any inaccuracy in machine movement.

Typical values are 0.5 to 1 millimeters.

Homing, search speed
Homing, clearance speed

When searching for a home switch, *WinPC-NC* starts moving an axis at the *search* speed. Movement stops when the homing switch trips. Movement then proceeds in the opposite direction at the *clearance* speed until the homing switch ceases tripping.
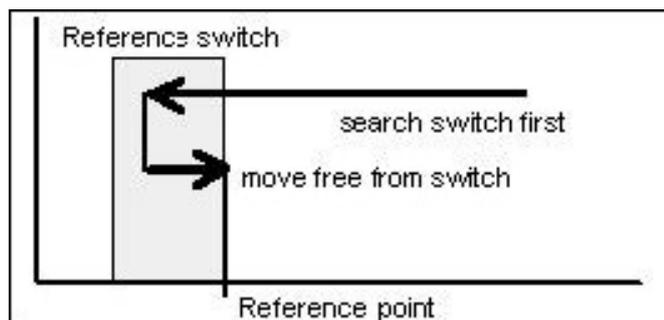
The point where the homing switch ceases to trip is the home position for the axis. A **slow** clearance speed is necessary to ensure the shortest possible ramp and the most accurate reference position determination.

Homing sequence

The home position determination process cycles through each axis searching for a home switch. This setting determines the order that the axes are searched.

The most common search sequence starts with the Z-axis, moving it upwards first, to ensure that the Z-axis is clear of any fixtures or a work piece. Other sequence orders may be applicable, depending on the machine.

Homing 4th axis

The 4th axis homing process can be performed before or after the other axes.

Checking home switches prior to machine initialization

homing with inactive switches only

When this checkbox is checked, **WinPC-NC** will refuse to initialize the machine until all home and limit switches are not tripping.

Some machines do not have individually wired home and limit switches (the home and limit switches wired together). This makes it impossible for **WinPC-NC** unambiguously to determine its position (e.g. which axis is reporting a limit or home trip signal) and begin a machine initialization.

Jog the machine to a location that is far from all ends, then begin machine initialization.

5.2.8. Basic settings - Jog
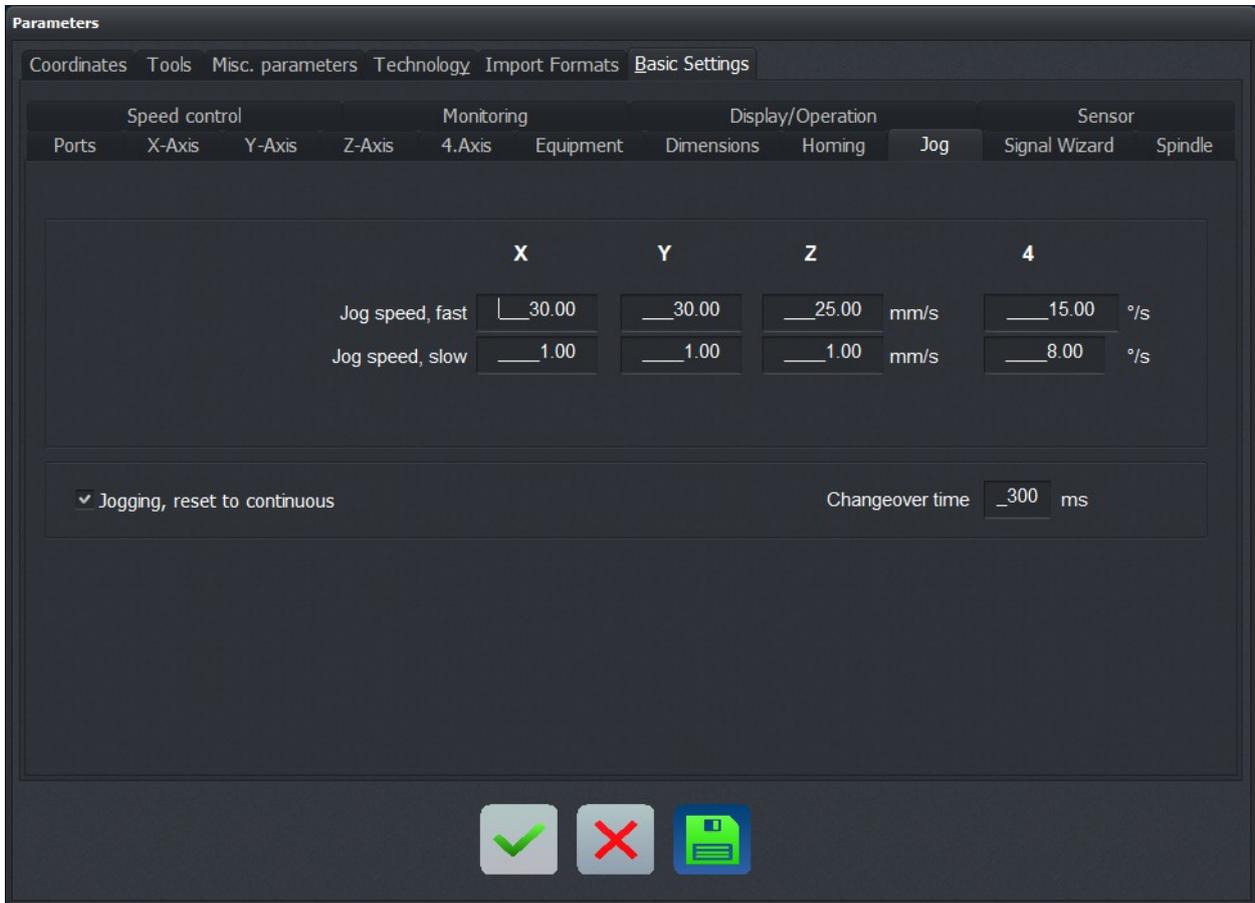
Parameter-Basic settings-Jog

Manual speeds - fast and slow

Specifies the jogging speeds – fast and slow.

Jogging, reset to endless

In jog mode, all axes can either be moved endlessly or discrete by distances which are selected by the drop-down menu.

The setting endless should be re-established automatically after each movement in order to avoid starting a pre-set distance inadvertently and to guarantee infinite move.

Change over time

Specifies the time, in milliseconds, to delay when transitioning from single step to continuous movement in jog mode.

When a key is hit briefly or a mouse button is hit, **WinPC-NC** outputs individual steps. If the key or mouse button is held for more than this time, motion switches to continuous mode and stays this way until the key or mouse button is released.

5.2.9. Basic settings - Signal Wizard
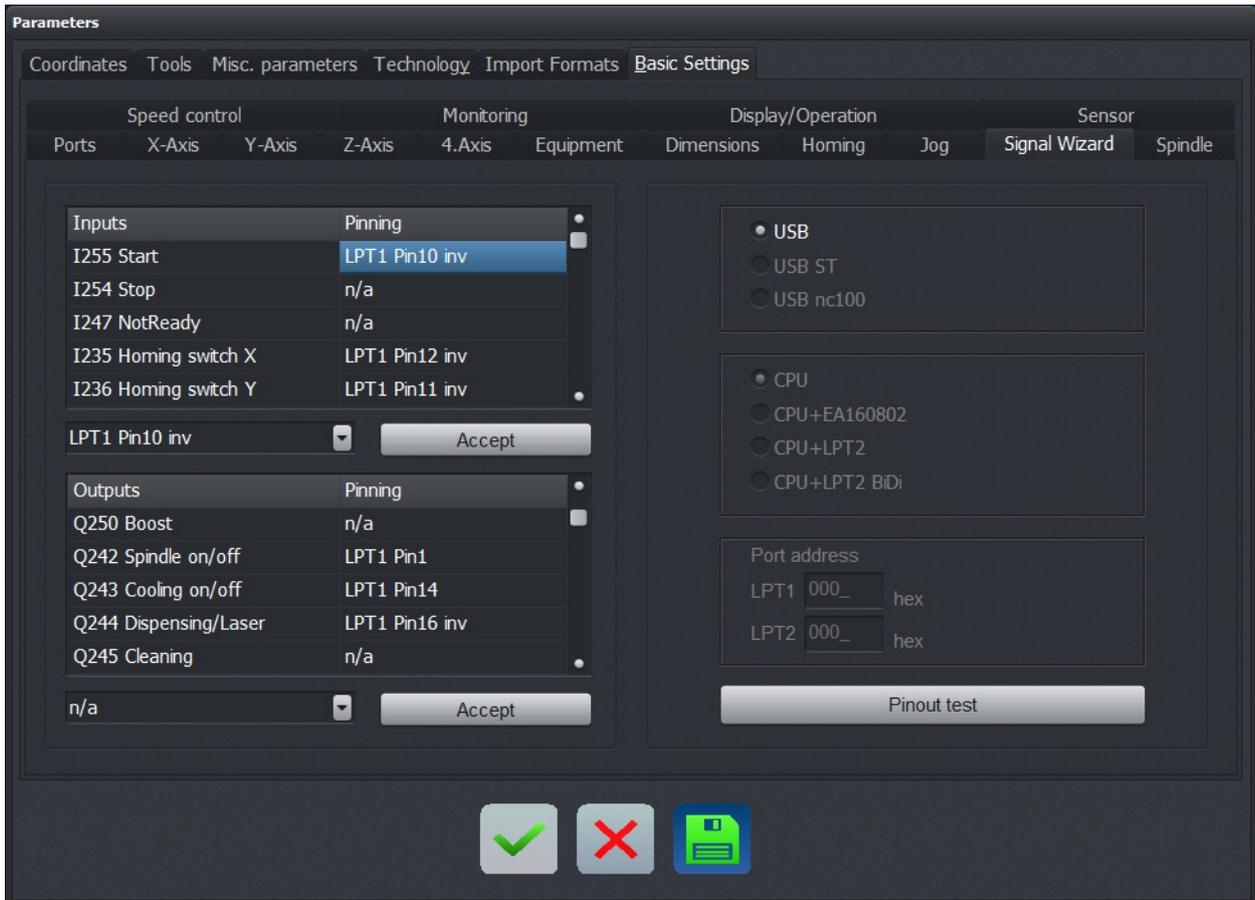
Parameter-Basic Settings-Signal Wizard

User-friendly definition of the signals

The *Signal Wizard* is a user-friendly way to define the necessary inputs and outputs. Setting up these signals can be a complex task. A chapter dedicated to setting up a machine using the *Signal Wizard* may be found later in this manual.

5.2.10. Basic settings - Spindle
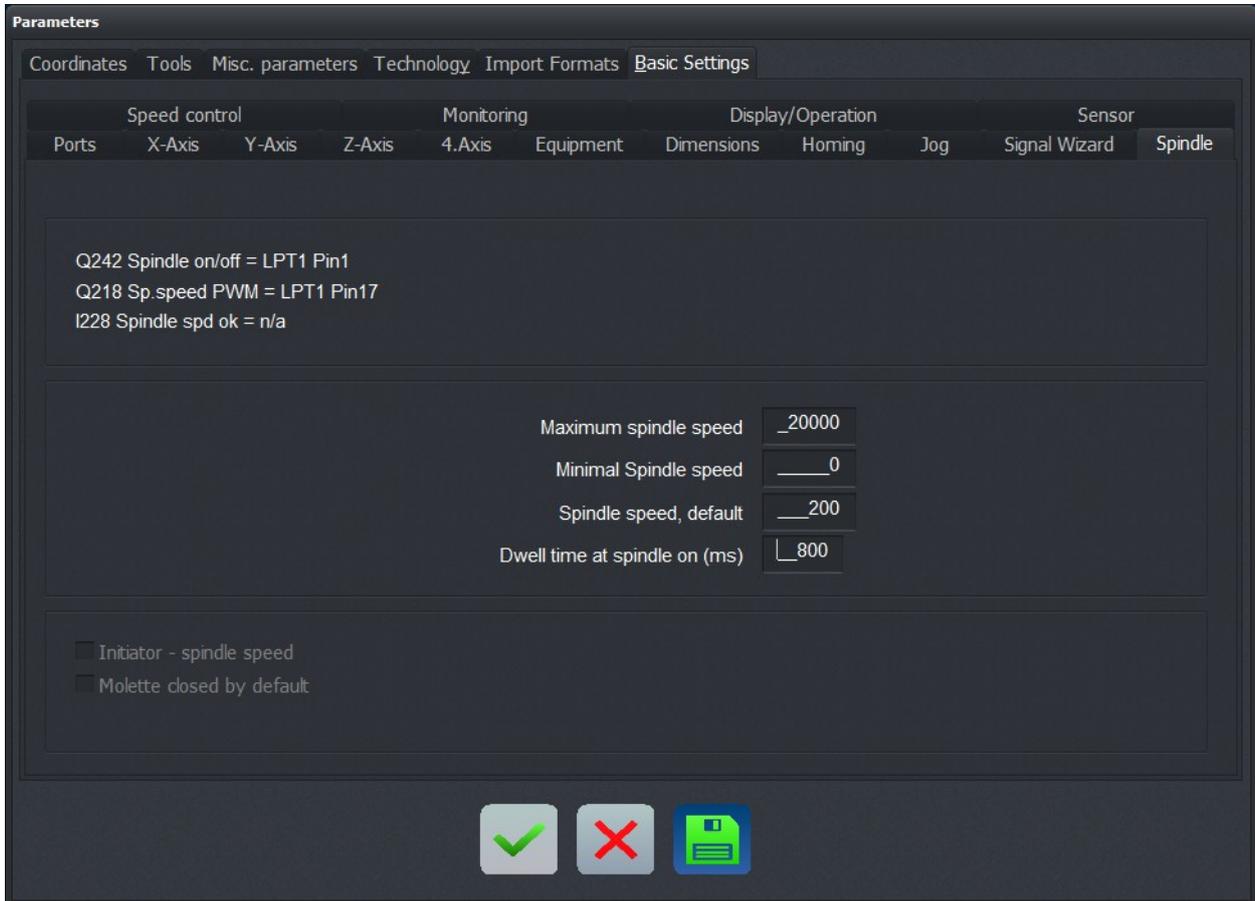
Parameter-Basic settings spindle

The upper section of the window displays the input and output signals associated with spindle control… a reminder to check that the spindle signals have been properly assigned.

If necessary hardware is available, *WinPC-NC* can control the rotation speed of a spindle.

The spindle speed is encoded as an 8-bit step number. Step 0 specifies that the spindle is off, step 255 specifies that spindle is running at its maximum speed. The step number is output, encoded, via one of 3 methods:

1.   Pulse Width Modulation (PWM)
        Output via pin 14 or 17 of nc100/ncUSB LPT1/axes controller.

2.   Binary number
        Output via pins 2 through 9 of ncUSB LPT2

3.   Analog output 0-10V at axes controller with optional input/output addon card.

Win-PC-NC is able to control a drilling or milling spindle in rotational speed. It is possible to define an individual speed for each tool or speed data can be obtained from the NC files.

Maximum spindle speed

Specifies the maximum speed of the spindle in revolutions per minute. Check with your NC machine and/or spindle vendor for this information.

Minimum spindle speed

Specifies the minimum speed of the spindle in revolutions per minute which is used with smallest output value. Check with your NC machine and/or spindle vendor for this information.

In most cases spindles cannot turn from 0 U/min on but have a minimum speed like 1000 U/min.

Default spindle speed

Specifies the default speed of the spindle in revolutions per minute. The default spindle speed is used whenever no spindle speed information is specified in an NC file or is associated with a tool.

Dwell time after spindle on

Dwell time after spindle on   Specifies the time, in milliseconds, to wait for the spindle to reach its specified speed when it is first powered on or after a tool change.

This ensures that there is enough time for the drilling spindle to run up prior to be used for the first time.

By another setting you can determine whether it is required to wait for a signal of the spindle control unit for displaying the final required speed. Many frequency converters and speed controllers provide such a signal and, if connected up and assigned to the correct pin, it is advisable to use this kind of monitoring.

Molette (chuck) closed by default

When *WinPC-NC* is used with an Automatic Tool Changer (ATC), a spindle with an electrically or pneumatically operated chuck is

required. Checking this checkbox ensures that the chuck is closed when the machine is initialized.

WinPC-NC can then maintain knowledge of the last used tool from session to session.

Sensor - Spindle speed

Signal for reached spindle speed
If your spindle exports an "at speed" signal and the necessary hardware is available, it is highly recommended that WinPC-NC be configured to wait for this signal rather than use a dwell time.

Check with your NC machine and/or spindle vendor concerning the "at speed" signal information.

5.2.11. Basic settings - Speed control
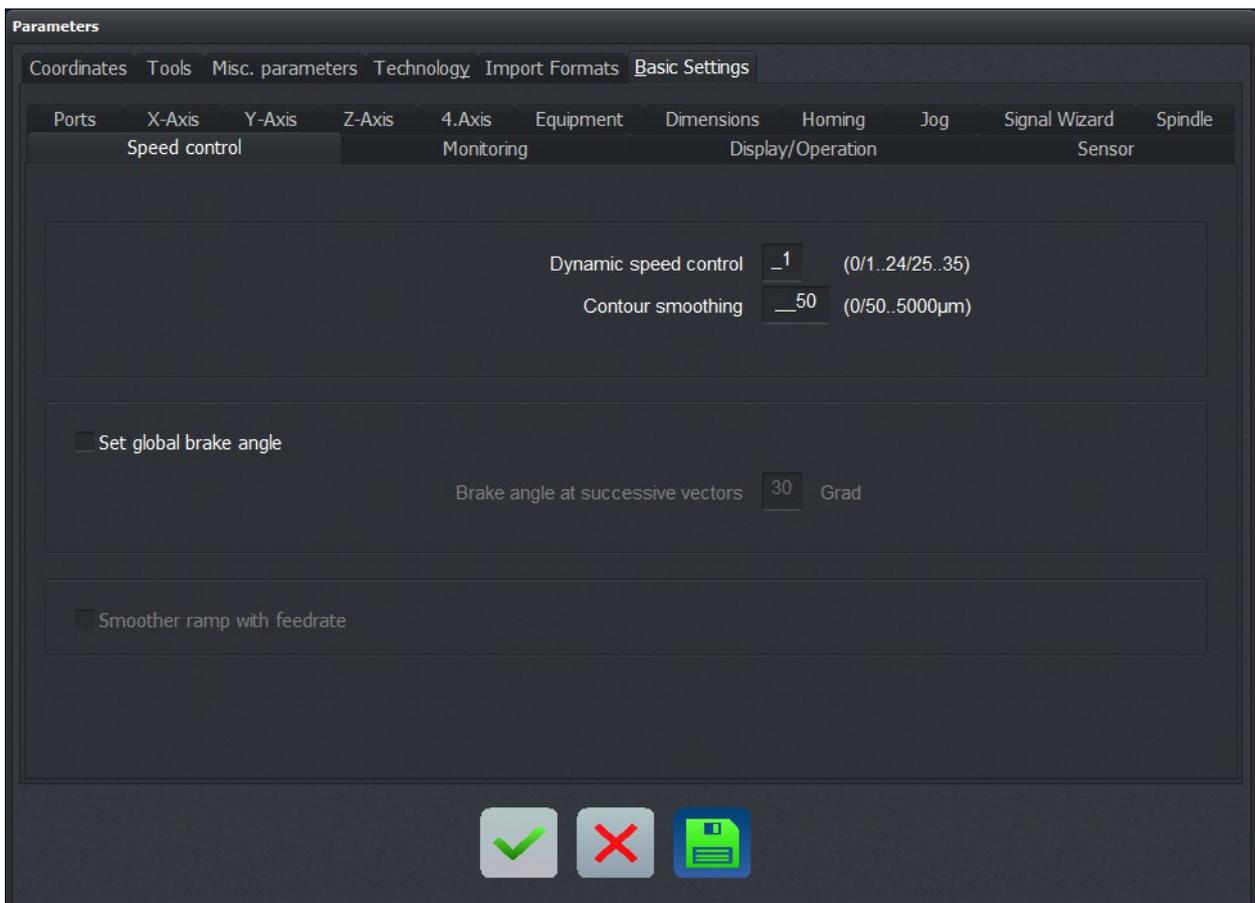
Parameter-Basic settings Dynamic speed control

Dynamic speed control

The speed control functionality of **WinPC-NC** attempts to recognize future contouring ("look ahead") and dynamically to adjust the cutting speed accordingly. The results are smoother edges and tighter corners.

look ahead function
How well speed adaptation can improve a job strongly depends on the mechanical properties of your machine and spindle. Optimal settings must be determined by experiment.

The speed control parameter controls how rapidly the machine will decelerate when it encounters small radius features. A value of zero deactivates speed adaptation; only the action of the set brake angle has an effect.

The values 25..35 use a totally different algorithm for the look-ahead function and controlling is made in 11 diverse modes to reduce speed in small arcs more or less. We recommend value 30 for the first trial.

Contour smoothing

Filter for closer control
The contour smoothing parameter specifies how aggressively **WinPC-NC** will attempt to convert a sequence of small vectors into a smooth curve. A value of zero deactivates contour smoothing.

The parameter defines a factor of 0..2000 which sets the length of the composed lines in µm.

Slow and precise milling motions should not exceed the value of 50 – 100. With rapid travels, e. g cutting or dosing, the vector length may exceed this value up to 300 – 1000.

This value affects essentially the number of produced vector data and operation speed.

Global determination of deceleration angle

If this parameter is activated, the angle in "*deceleration angle with successive vectors*" affects all tools and is no longer defined separately in tool parameters.

5.2.12. Basic settings - Monitoring
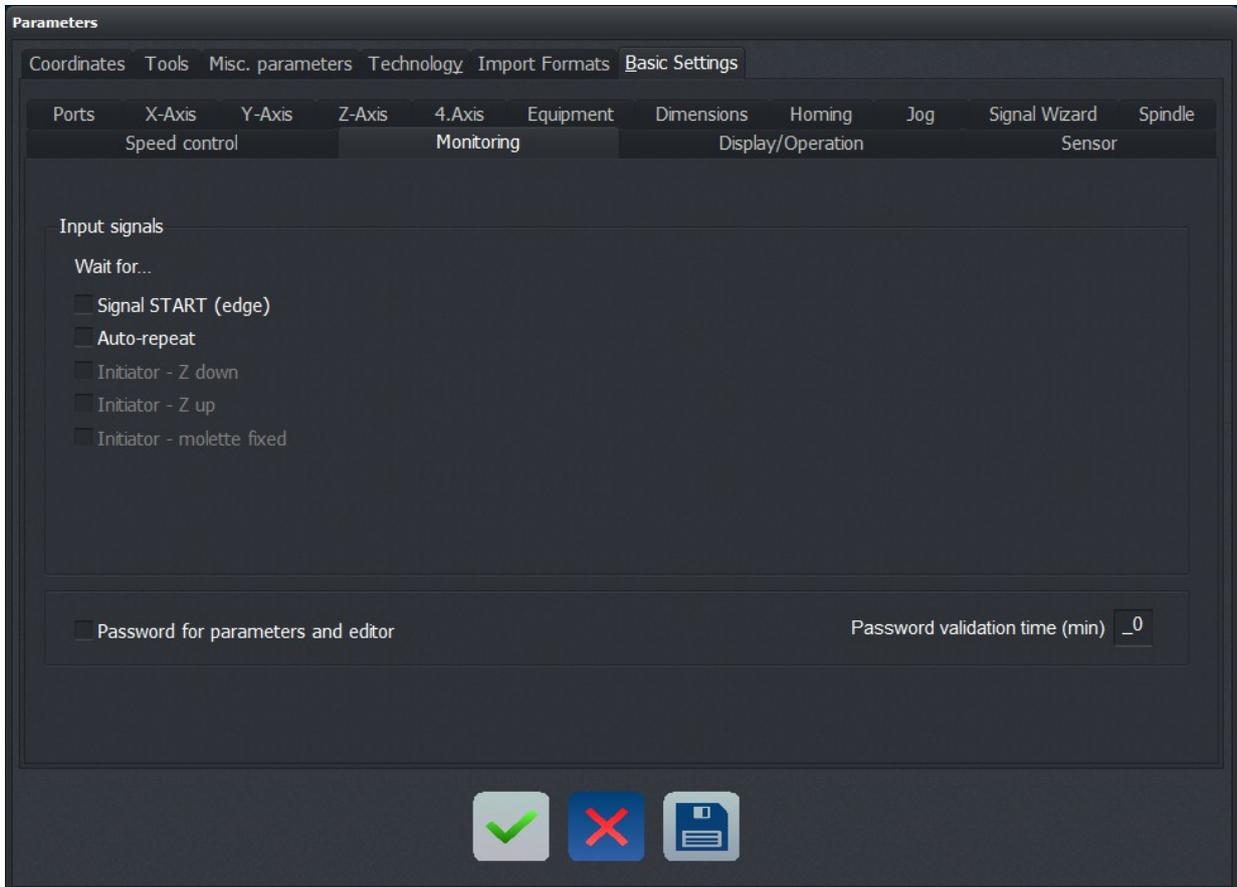
Parameter-Basic settings Monitoring input signals

All input signals and their associated parameters are grouped to-gether.

Input signals

Synchronisation with various sensor signals

WinPC-NC can synchronize the current job with various input signals. Check with your NC machine and/or spindle vendor concerning which signals may be available.

The signals and their definitions:

Start signal   **WinPC-NC**, after the current job been started, waits until this signal ("external hardware is ready"; "you may start") to transition from LOW to HIGH before beginning processing.

Sensor Z down	*WinPC-NC* waits until this signal ("Z-axis is down") to transition from LOW to HIGH before continuing processing.
Sensor Z up	*WinPC-NC* waits until this signal ("Z-axis is up") to transition from LOW to HIGH before continuing processing.
Clamping	After the current job has been started, *WinPC-NC* waits for this signal ("work piece clamping is activated") to transition from LOW to HIGH before starting processing.
Spindle speed	**WinPC-NC** waits until this signal ("spindle has reach specified speed") to transition from LOW to HIGH before continuing processing.

Automatic repeat

Automatic repeat of a process

When this checkbox is checked, *WinPC-NC* will check for the *Start* signal, and if it is HIGH, restart the currently loaded job. No keyboard or mouse input is required.

Password and period of validity

This parameter specifies a password which must be entered prior to changing any parameters or a loading an NC file.

The period of validity specifies how long the password will remain in effect.

5.2.13. Basic settings - Warning messages

Warning !

Only available in *WinPC-NC Professional*.
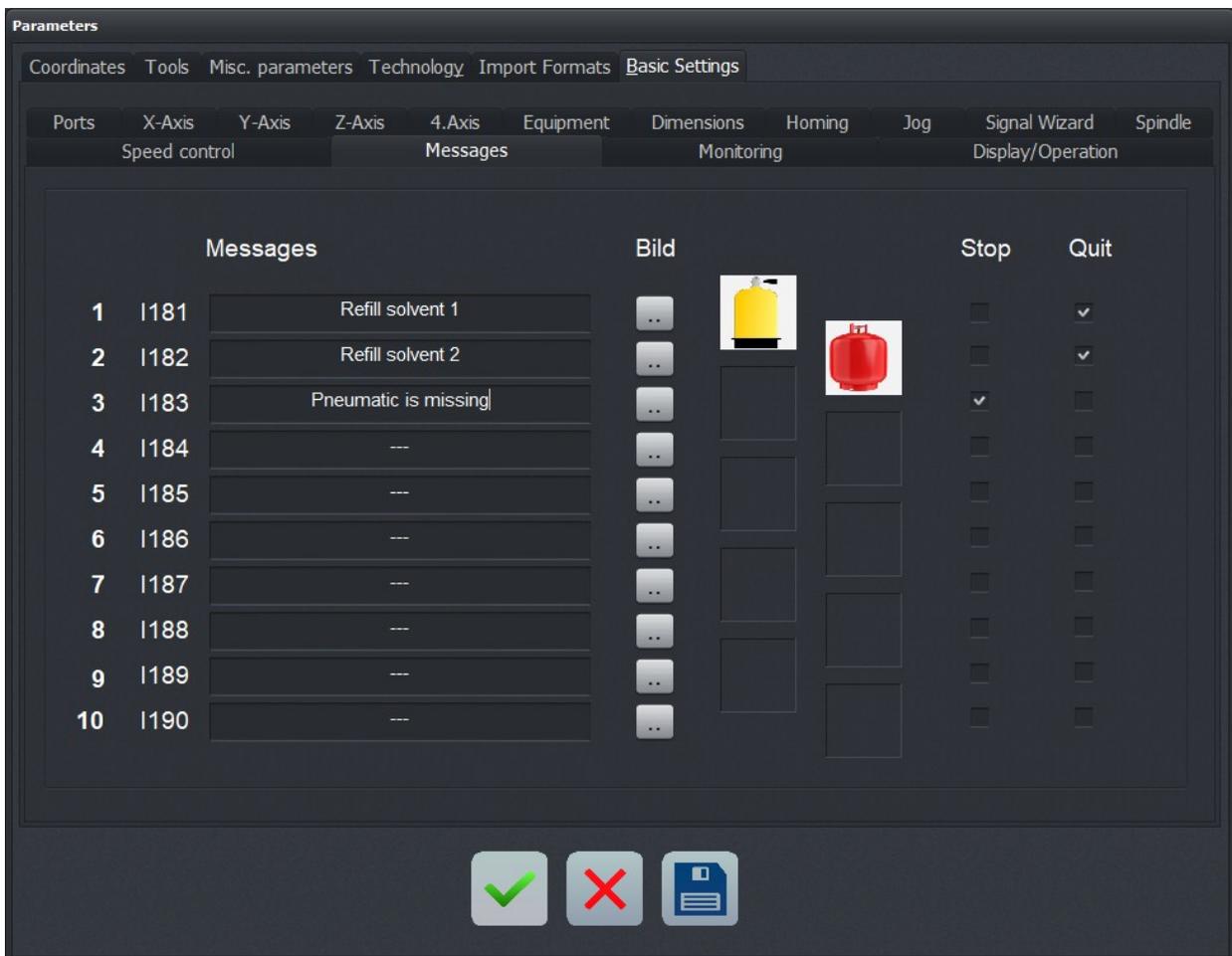
Synchronize warn-and error messages with input signals

WinPC-NC Professional can freely define various monitoring functions and assign them to specific input signals.

Assignment of messages and pictures to error signals	For this assignment the inputs I181-I190 Error signal 1-10 are available and have to be defined by the signal wizzard prior to use. In the following step any text or picture can be assigned to any signal and it is possible to stipulate the method of monitoring.

By the additional available switches *Stop* and *Quit* the method of monitoring and further operation process can be determined.

Stop activ*e* Constant monitoring of the error signal. In case of error the current job is interrupted.

Stop inactive Monitoring of the error signal when job starts.

Quit active Enables accepting the error status and continuation of the operation process without rectifying the error itself.

Quit inactive Refuses continuation of further operation until the error has been rectified.

Message after detected input signal

5.2.14. Basic settings - Stand alone job execution

By using **WinPC-NC Professional** and the relevant axis controller it is possible to save the complete job process together with all its the operational parameters durably in the controller. So it is possible to execute the job again without the necessity of a connected computer or a running **WinPC-NC** system.

Mass production without constantly running system

This is a very useful function for mass production during an extended period and it does not require to be managed by a host computer.

However, for using this function the axis controller must be programmed correctly and specifically. On request we are gladly prepared to supply further details. Please do not hesitate to contact us in this matter.

Detailed information concerning saving a job is available in an additional HowTo document at our web site.

5.2.15. Basic settings - Sensor probe

All settings and defintions concerning edge probe, surface probe, tool length probe and touch plate are pooled in this dialog.
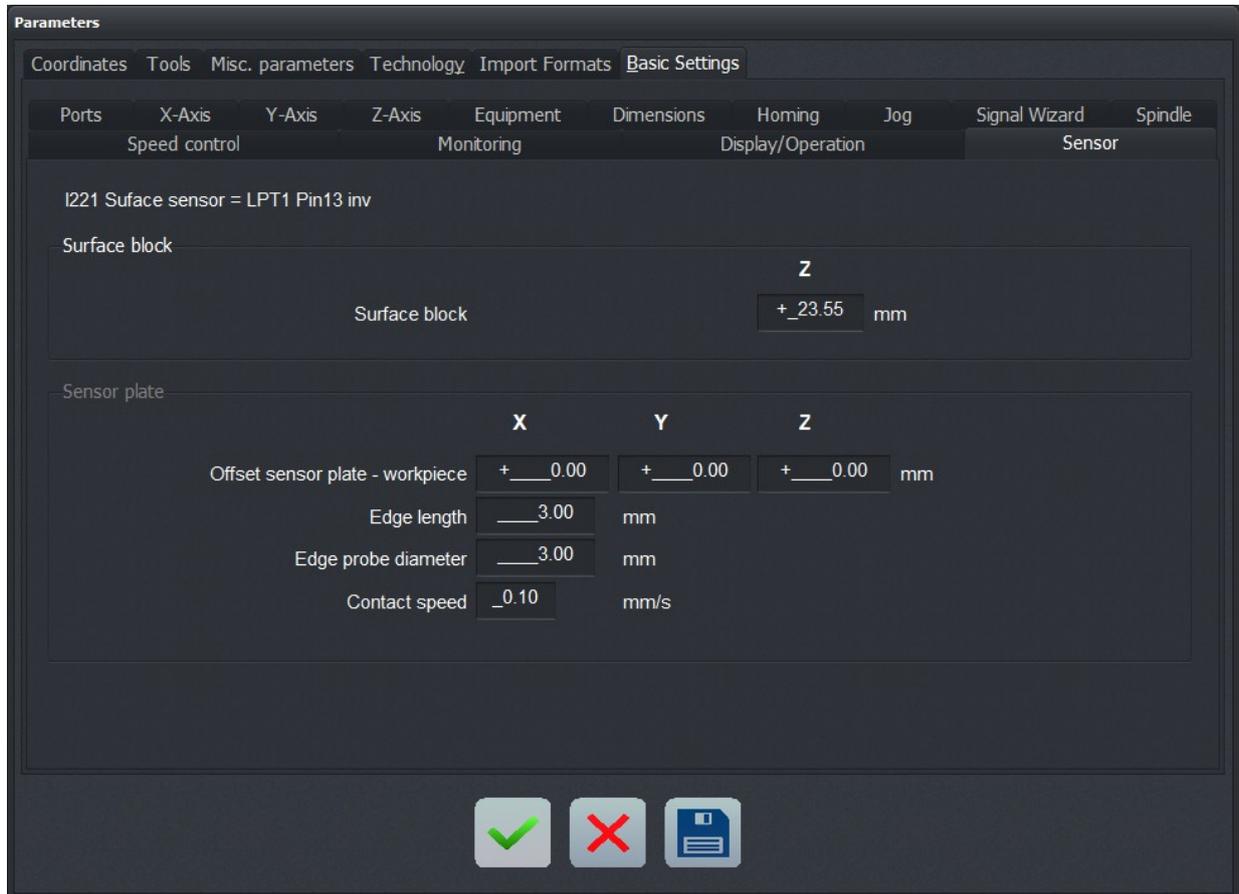
Parameter settings for different probes

Surface block

Automatically measure Z-axis heights

WinPC-NC can automatically measure work piece Z-axis heights using a surface block (a sensor). The sensor block outputs a signal when it is contacted from above, and the signal is processed as an input to **WinPC-NC**.
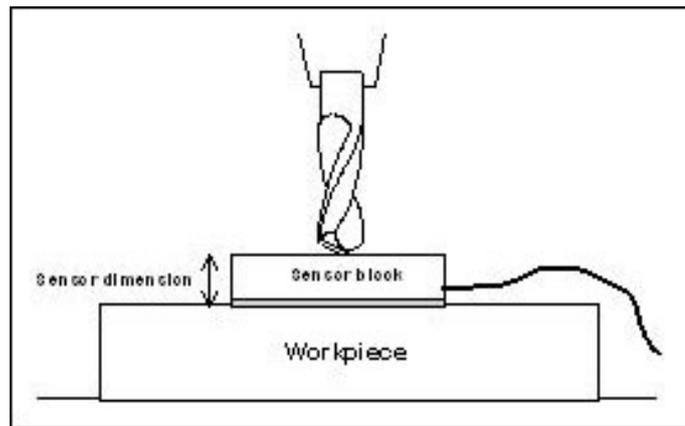
Automatic measurement of the Z-axis zero point

The measuring procedure involves several steps:

1.   Place the sensor block on the work piece surface or on the machine

2.   Jog the tool over the sensor block.

3.   Start the measurement using the MOVE-JOG menu function

4.   *WinPC-NC* slowly moves the tool down to the sensor block and stops when it makes contact. The position is checked and added to the defined sensor block height, the result being stored as a parameter.

Touch/Sensor plate

Detailed instructions for using a touch plate can be obtained in our homepage www.Lewetz.de in the *Help* menu – HowTo instructions.

Offset touch plate workpiece

The values correspond to the thickness of the stop angle in X and Y direction. The value Z complies with the thickness of the contact plate without stop angle. These values are used for computing the zero point after contact.

Edge length

The edge length indicates the length of the contact plate in X-direction. This value is used for measuring an angle. Hereby the angle is measured at both ends of the contact plate in order to obtain the most exact result.

Probe diameter

This value indicates the diameter of the milling or drilling tool which is used as probe This value is also required for computing the zero point.

Touch speed

In order to avoid inaccuracies in measurements and damages of the fixed probe, touch speed should not be too high.
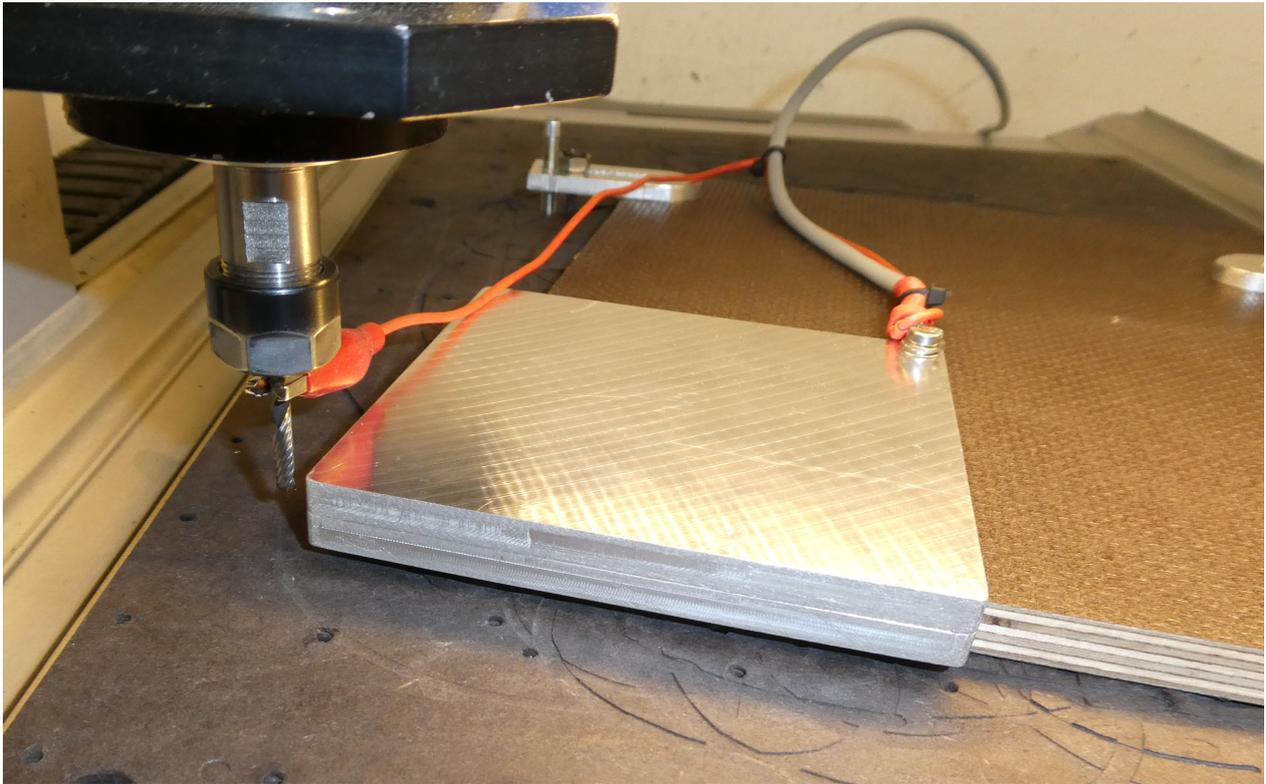
WinPC-NC: lateral approach to touch plate

5.3. Coordinates

5.3.1. Coordinates - Auxiliary points

Parameter-Coordinates

Park position

fixed position after job or for tool change

The park position is a convenient location to which the machine can be quickly moved and be parked. For example, a location where it is easy to manually change a tool or "out of the way" when placing stock in a clamp.

The park position can be selected as the target of a jog or can be used as part of a tool change operation.

The position is in absolute machine coordinates (distances relative to the home switches); units are in millimeters.

Scaling factors

The scaling factors make it possible to compensate for machine initialization difficulties. If the axis resolution and the unit of measurement are set correctly but the machine does not consistently move to exact position expected, a scaling factor can be used to-correct this issue.

Correct dimen-sons for engra-vings

Concerning engravings it is sometimes advisable or useful to adjust the dimensions to templates or carriers.. This can be easily done by scaling factors which can be individually defined for all axes.

These values are to be indicated by 3 digits behind decimal places and are used for multiplying coordinates to be moved to.

Tool lift

height for unproductive moves over material

The clearance distance specifies an additional height of the Z-axis above the zero point level. With each job process the tool is lifted above the zero point by this distance and the new height is used as tool lift height.

On the next plunge movement, *WinPC-NC* first covers the safety clearance down to the zero point at high speed, before pressing into the material with the defined plunge speed.

Zero point

The zero point refers to the origin of the coordinates of the work piece within an NC file. All distances are measured relative to this point.
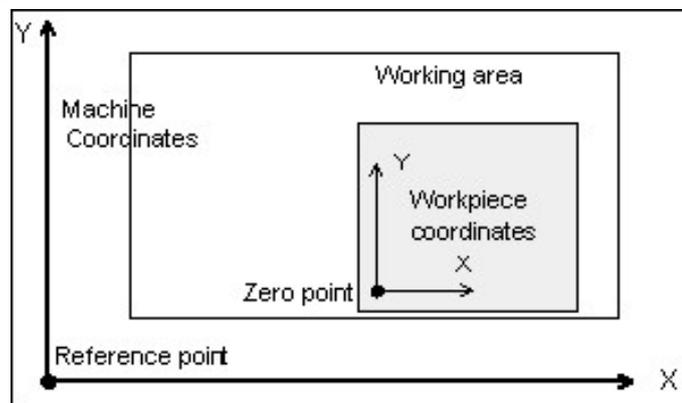
Workpiece and machine coordinates

zero point is always ithe origin of nc coordinates

The assignment is made by jogging to the desired position and saving the position as the zero point. A zero point can be specified for X, Y, Z, XY, and XYZ. Once a zero point has been assigned for X, Y, and Z, the job can begin.

The machine position is in **absolute machine coordinates** (distances relative to the home switches); units are in millimeters.

Zero point in file

The workpiece zero point is the point in the NC file and its position is defined in the coordinate parameter. However, it can also individually be placed inside or outside of the workpiece and the required positons are defined here.

Warning !

Using multiple zero points this parameter is standardly set to origin of the coordinate system and can not be changed.

Detailed information for using multiple zero points can be obtained in our homepage in the *Help* menu - *HowTo* - instructions.

Six positions are selectable :

Bottom left
The zero point is at the smallest X- and Y-axis coordinates in the file, normally at the bottom left edge. Use this choice when working with HPGL files.

Coordinate origin
The zero point is at the coordinate origin assign by the CAM program that produced the NC file. Use this choice when working with G-code files.

Center
The zero point is exactly in the center of the coordinate dimensions in the X- and Y-axis directions of the workpiece. This setting is useful for processing round work pieces (e.g. plates).

Bottom right
The zero point is positioned at the highest X and smallest Y coordinate of the file.

Middle right
The zero point is positioned at the highest X coordinate and exactly between the smallest and highest Y coordinate.

Top left
The zero point is positioned at the smallest X and the highest Y coordinate of the file.

Working piece area monitoring

*Software
area monitoring*

The working area defines the section, e. g. for marking material dimensions. Limits are visible in the graphical display and it is immediately recognizable whether milling or engraving can be effected with the material piece. A check is made when job starts.

Maxiimum plunge depth for Z-axis

The working area for the Z-axis determines the maximum plunge depth to which tools can move without damaging the bed of the machine.

Easy definition by moving to corners

It is very easy to move the machine to the lower left and upper right corner of the desired area by pressing the function keys F5 and F6.

These parameters are not to be mixed up with the values determining the size of the machine table and thereby the maximum moving area.

Measuring is made in millimeters and distances refer to the machine reference point (machine coordinates).

The management of an additional area is activated when you enable the corresponding work piece by checkbox.

Homing 4th axis

The 4th axis homing process can be performed before or after the other axes.

Zero point 4. axis

In this area you have to set the zero point for the 4th axis. Depending on the mechanics used for controlling the 4th axis, it is about a position in mm or degrees and is accurately moved by the 4th axis in home running.

5.3.2. Coordinates - Multiple zero points

If several zero points are activated, coordinates of the various zero points can be displayed or edited in the relevant tab.

Parameter

Koordinaten Werkzeuge Sonstige Technologie Makro Datenformat Grundeinstellungen

Hilfspunkte Nullpunkte

	X	Y	Z	
Nullpunkt 1 (G54)	+___52.12	+___56.00	+____0.00	mm
Nullpunkt 2 (G55)	+___96.41	+___56.00	+___20.40	mm
Nullpunkt 3 (G56)	+__169.45	+___74.13	+____0.00	mm
Nullpunkt 4 (G57)	+___10.98	+__147.36	+___56.30	mm
Nullpunkt 5 (G58)	+____0.00	+____0.00	+____0.00	mm
Nullpunkt 6 (G59)	+____0.00	+____0.00	+____0.00	mm

4.Achse

Nullpunkt 1 (G54)	+__180.00	mm	Nullpunkt 4 (G57)	+__310.58	mm
Nullpunkt 2 (G55)	+___52.74	mm	Nullpunkt 5 (G58)	+____0.00	mm
Nullpunkt 3 (G56)	+___96.03	mm	Nullpunkt 6 (G59)	+____0.00	mm

Parameter-Coordinates - Multiple zero points

Select zero points with G54 to G59 The different zero points can be selected or activated either in the NC program or by buttons G54 to G59.

5.4. Tools

Project-related tool management

WinPC-NC has a user-friendly tool management function. It is possible to manage up to 10 (30) tools - each with unique parameters - for each project.

The parameters are divided between several dialog boxes. **WinPC-NC** stores all the values defined in these dialog boxes into the tool file using the PARAMETERS-SAVE menu function. Tool files have the *.WPW filename extension.

5.4.1. Tools - Activation

If 30 tools are activated in parameter – basic settings – equipment, buttons are displayed for selecting the specific tools.

Tools1-10, 11-20 and 21-30
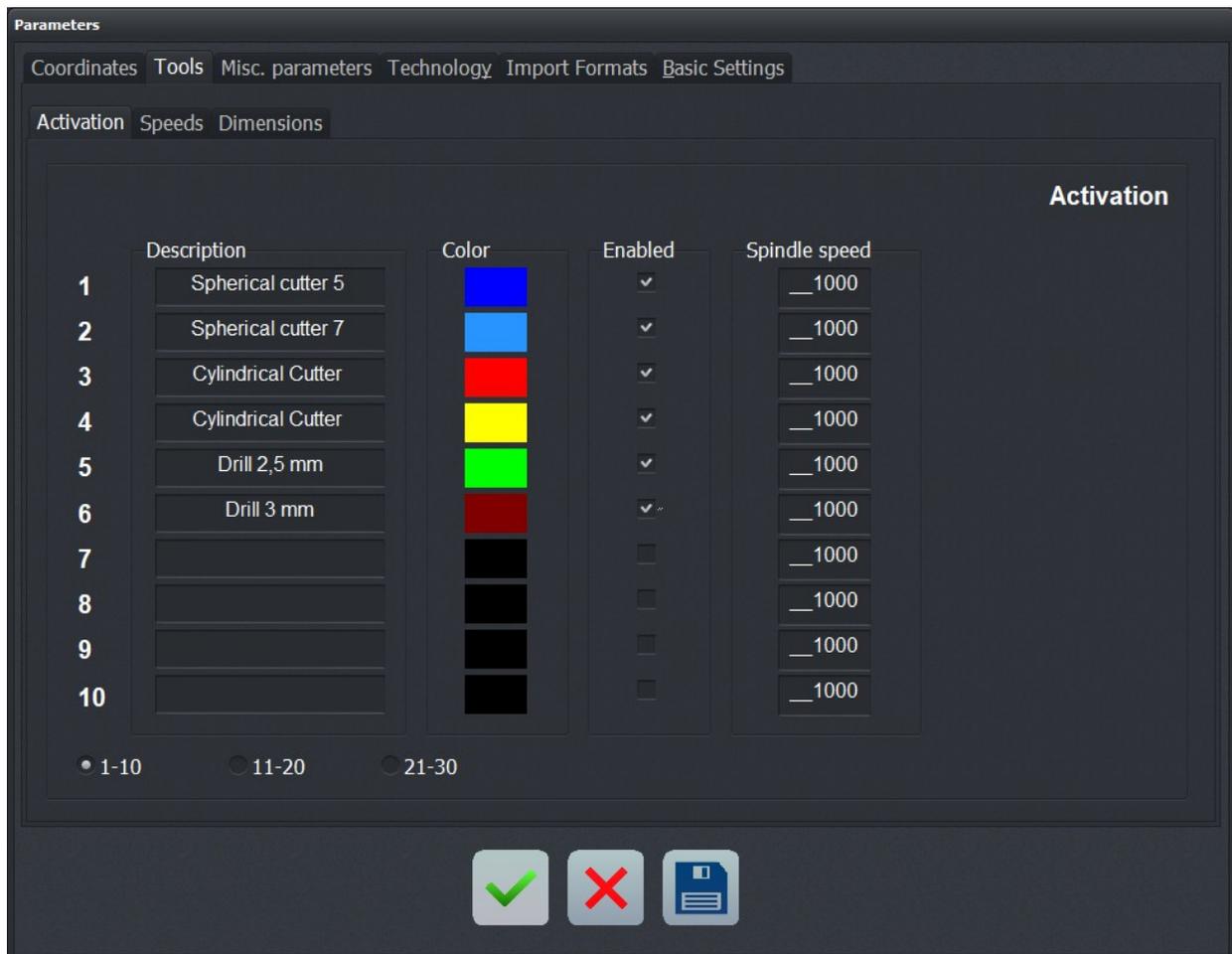
Parameter-Tools activation

Description

Descriptive tool names

Each tool is identified with a name which is used when a tool change prompt appears.

Color

Colors for graphical view

Clicking the color box associated with each tool opens a color selection dialog. Choose a color that suits your requirements or harmonizes with the CAD program you are using. This is useful for those with color blindness as they can specify a color palete that works best for their specific difficulties.

The dialog window for selecting a new colour opens after clicking the relevant colour area.

Enabled

Activate tools individually

Each tool can be individually enabled or disabled. When a tool is disabled any command involving it will be ignored and any tool-path involving it will not be shown in the *Display Area.*
Tools can also be enabled and disabled via the numbered tool area at the left of the *Tool Bar*.

With loaded NC files, *WinPC-NC* is able to recognize automatically which tools are used and modifications and settings can solely be done for these tools. Any other tools and those which are not used are inactive in these dialogs

---

Spindle speed

Defines the spindle speed for the specified tool if...

- the spindle speed is not be specified from within an NC file

- *WinPC-NC* is directed to ignore the NC file spindle speed directives.

5.4.2.   Tools - Speeds

Parameters			

Coordinates Tools Misc. parameters Technology Import Formats Basic Settings

Activation Speeds Dimensions

Speeds

	V-Plunge	V-Advance	V-Withdrawal	Brake angle
1	0.01	30.00	10.00	30
2	1.00	30.00	5.00	30
3	1.00	25.00	5.00	30
4	2.50	10.00	3.00	50
5	1.00	1.00	1.00	50
6	5.00	20.00	1.00	50
7	5.00	1.00	5.00	30
8	5.00	1.00	5.00	30
9	1.00	1.00	1.00	30
10	1.00	1.00	1.00	30

● 1-10   ○ 11-20   ○ 21-30

ParamParameter-Tools Speeds

Plunge speed

Specifies the speed that each tool is pushed (plunged) into the work piece.

This number is dependent on the material being used, the tool involved, and the mechanical properties of the machine.

Advance speed (feed rate)

Specifies the speed that each tool is pushed (fed) across the work piece. This value is not used in straight forward (hole only) drilling applications.

When *WinPC-NC* is used for milling, engraving or grinding, this number is dependent on the material being used, the tool involved, and the mechanical properties of the machine.

Withdrawal speed

Specifies the speed that a tool with withdrawn (raised) from the work piece.

Brake angle

Specifies the maximum angle, in degrees, between two linear movements where movement can (still) take place at full speed

look-ahead speed control   If the angle between the current motion and the next is less than the break angle, no deceleration is necessary - the constant speed phase can continue.

An example of how this works can be seen in movement around a circle. A circle is approximated via a large number of individual vectors. The angle between any two of the vectors is very small. As a result, the circle can be completely traversed at maximum speed.
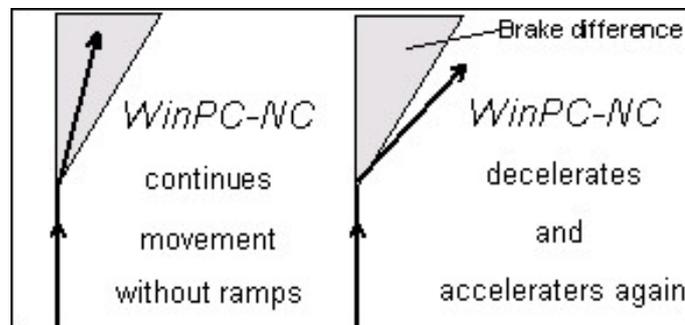
Brake angle

A break in movement is performed before and after each tool movement.

This parameter is not used in straight forward (hole only) drilling applications.

5.4.3.   Tools - Dimensions

Parameter-Tools Dimension

First Cut Depth

Specifies the depth of the first cut from the Z-zero point. Additional cutting depth is handled separately, via the # Additional cuts and the Additional Cut Depth parameters (see below).
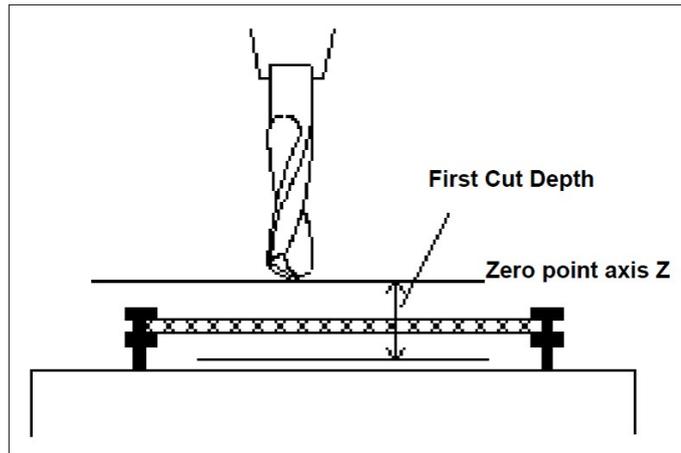
First cut depth measured from the zero point of the Z-axis

Additional cuts and Additional Cut Depth

Milling step by step

Milling thick or hard materials often requires multiple sessions. In order to avoid starting the process several times consecutively by changed insertion depths both parameters "*number of additional cuts*" and "*additional deeper cut*" are to be used.

The number of additional steps are executed after the first cut. Each time a cut is made deeper from the home position of the previous cut by the relevant value '*Depth of additional cuts*' .
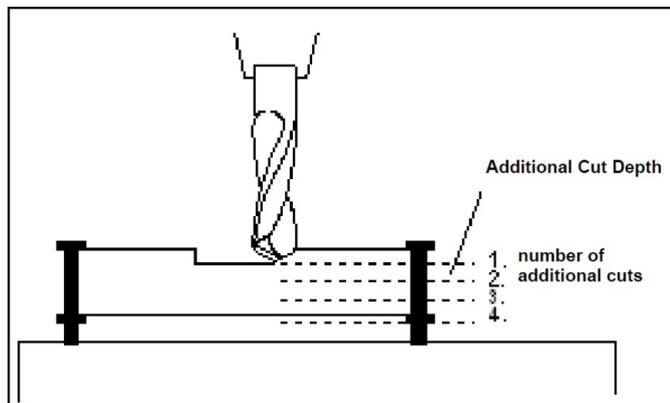
Additional cuts and Additional Cut Depth

5.4.4.   Tools - Length measurement and compensation

These setting are only visible if the tool length measurement and compensation functions are activated in *Basic Settings-Equipment*.
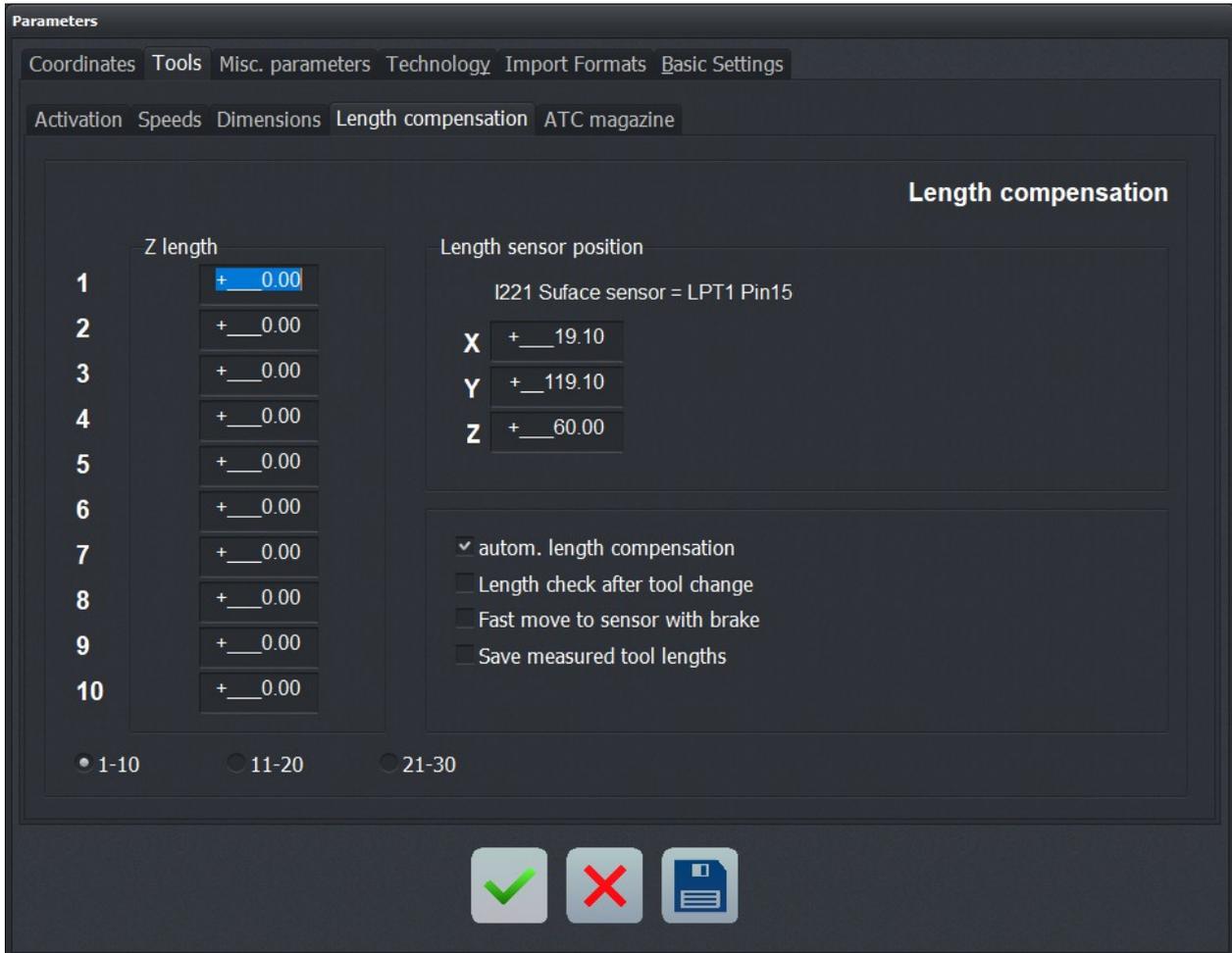
Parameter-Tools-Tool length measurement and compensation

Z-axis length

The length of a tool is specified here. Normally, these boxes cannot be edited; the values are registered automatically when tools are measured. The length difference used by the compensation function is calculated on the basis of these values.

Sensor position

length sensor/probe

The tool length sensor is a switch, button, pressure plate, solid state contact sensor, or other device which transitions, unambiguously upon tool contact.

This parameter defines the position of the tool length sensor in absolute machine coordinates (distances relative to the home switches) and a Z-axis height; units are millimeters. To measure a tool, *WinPC-NC* moves to this position and Z height, then lowers the tool slowly until the sensor transitions (off-on or on-off).
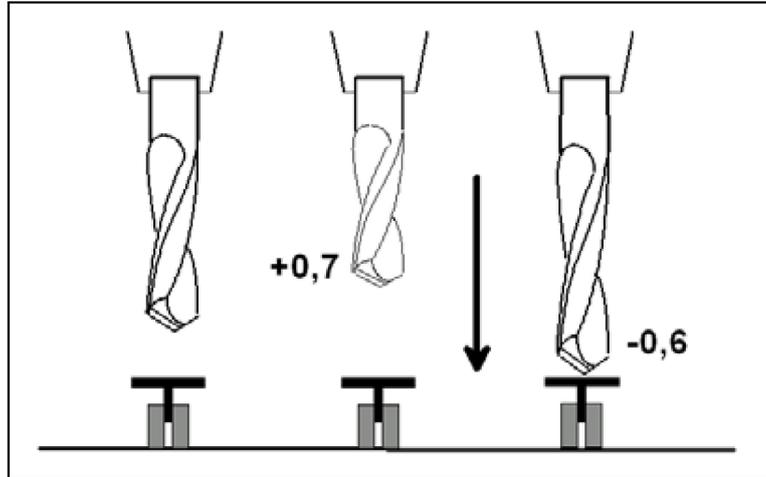
Schematic sketch: length sensor and automatic tool measuring

Automatic length compensation

This parameter enables the automatic tool length compensation. Due to the release of a separate parameter, compensation can be solely used for projects with the need.

Tools are either measured automatically after tool change or explicitly for each required tool by MEASURE TOOL function.

Length check after tool change

Enable/disable tool length compensation after a tool change. By default, *WinPC-NC* does not perform automatic tool length compensation after a tool change.

Fast move to sensor with brake

The automatic tool length compensation process can be sped up by enabling this feature. When enabled, *WinPC-NC* will move to the sensor at the fastest possible speed, accelerating and decelerating as necessary.

Save measured tool lengths

WinPC-NC will save all measured tool lengths in the parameter file when this feature is enabled. In applications that constantly use the same tools, this can speed things up - the tools do not have to be measured again.

5.4.5. Tools - ATC magazine

This setting is only visible when the Automatic Tool Changer (ATC) capability is enabled in *Basic Settings-Equipment*.

Parameter-Tools-ATC Magazine

Control automatic tool changer

WinPC-NC can control an Automatic Tool Changer (ATC) with up to 10 (30) magazine positions.

A spindle with an electrically or pneumatically operated chuck is required when using an ATC. This chuck must be controlled via input/output controls.

The position of each magazine location should be specified at a short distance to the actual pick up location. Macros should be used to handle the exact sequence of picking up and releasing tools as well as handling the final movements, waiting times, and any input/output controls.

The macro procedure is described in a subsequent chapter.

Moving tool magazines (e.g. retractable cabinets, circular magazines) require 4th axis control only available in *WinPC-NC Professional*.

Parameters for tool changer are only available with enabled ATC.

With *WinPC-NC Professional* it is possible to monitor individual magazine positions and collets by additional sensors. Thus accidental tool collisions can be avoided.

Furthermore it is solely a special feature of WinPC-NC Professional to use a 4th axis for tool changer, e. g. retractable drawer or turret magazines.

No Z lift at ATC

By default, *WinPC-NC* does a lift to the maximum Z height before moving to the appropriate magazine location.

Enabling this feature will disable the maximum lift, saving time.

CAUTION: Enabling this feature may increase the chance of a collision with a fixture or other components.

No move back to last position

Enable/disable remembering the last position before a tool change. Moving to a tool changer magazine position, *WinPC-NC* will remember the previous position and return to it after a tool change. This often results in unnecessary movements and may not be desirable.

5.5. Miscellaneous parameters

Settings which apply to a job process or *WinPC-NC* operation.

5.5.1. Miscellaneous - Display/Operation
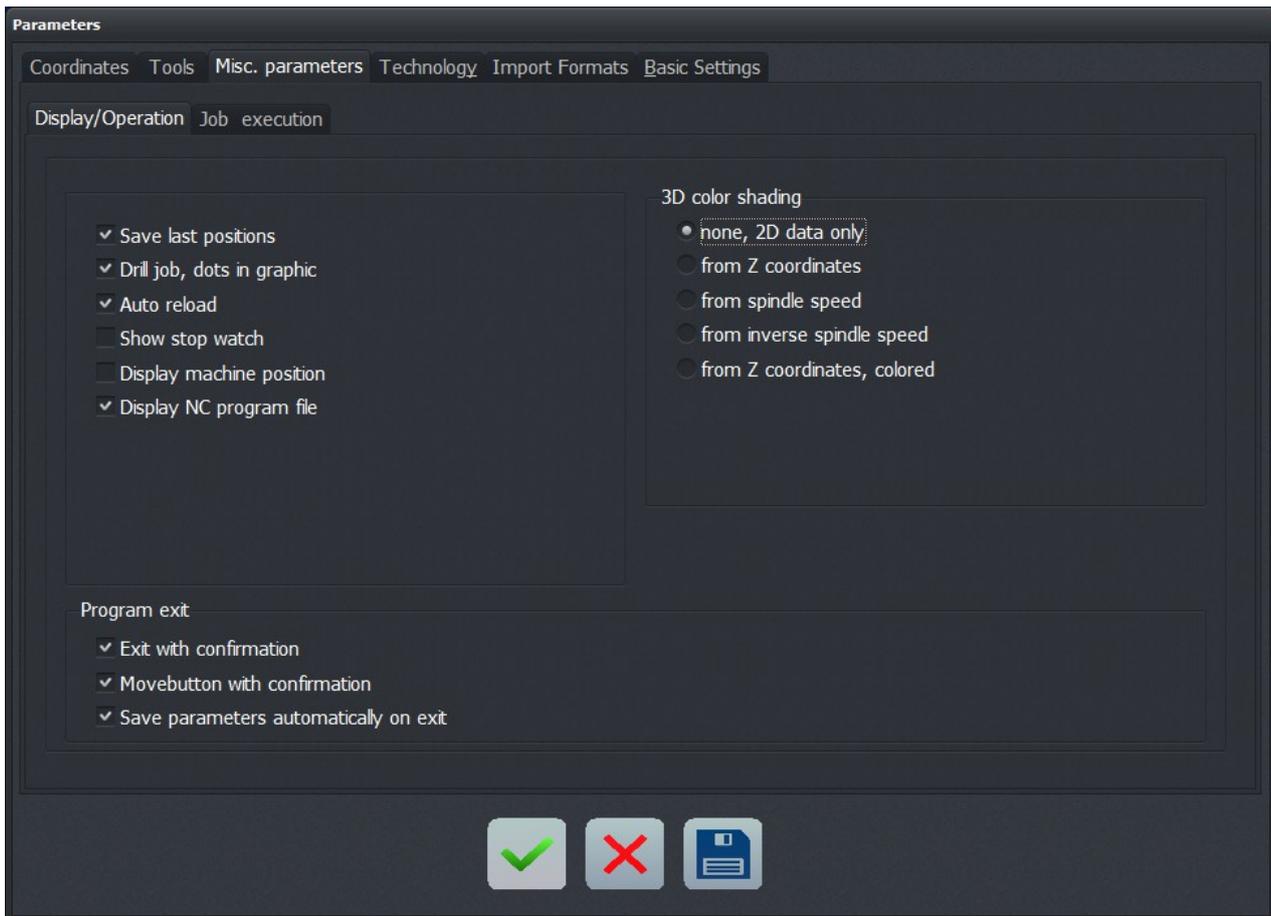
Parameter-Misc. parameters-Display/Operation

Drill job, dots in graphic

Mark drill holes	By default, the *Display Area* does not show the location of any drill holes. Activating this parameter places a small circle at the hole insertion points for HPGL, MultiCAM, or G-code files.
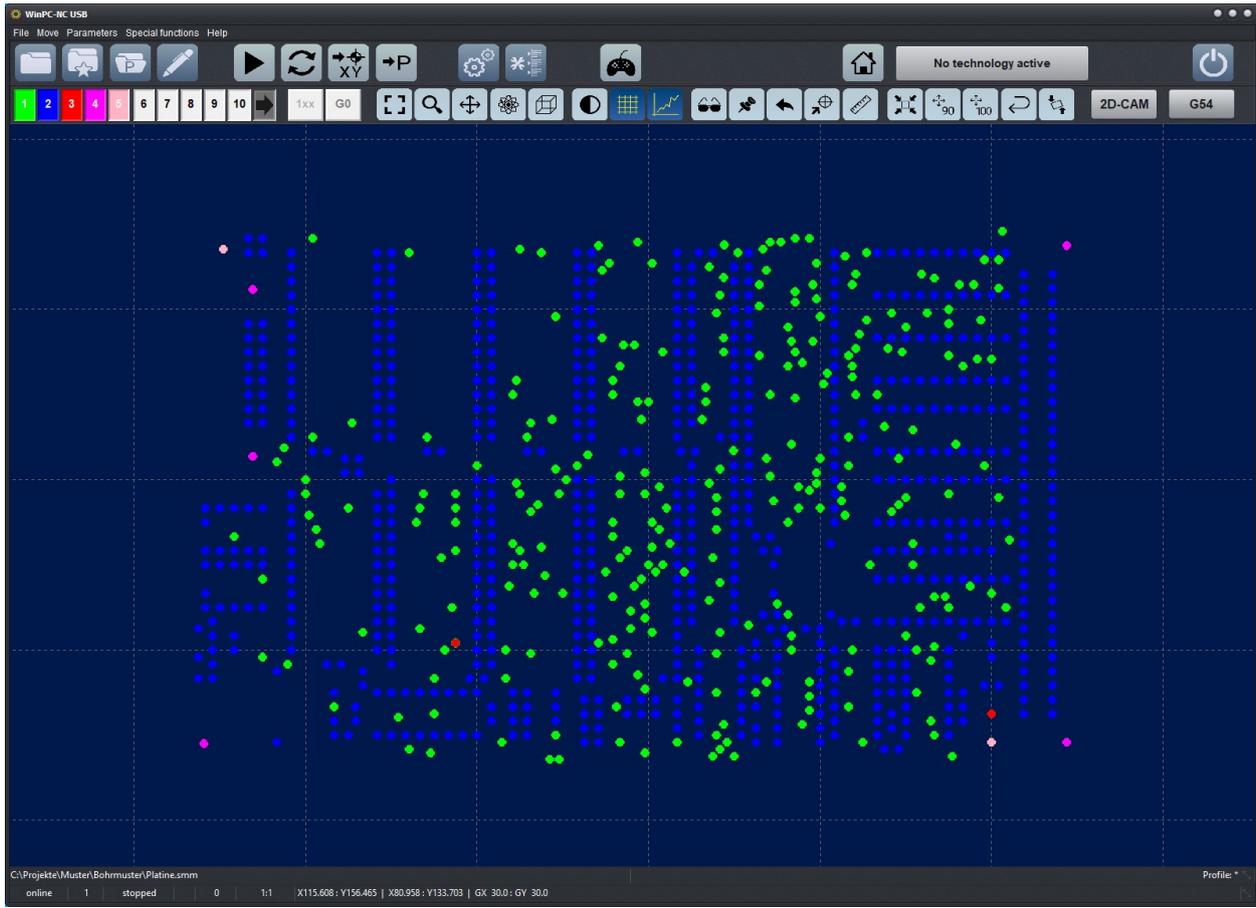
Drilling job with hole locations indicated

Automatic reload

When this feature is enabled, *WinPC-NC* continuously monitors the currently loaded file and reloads it whenever the file modification time is newer than that of the copy currently loaded.

It is possible to edit an NC file in the drawing program, switch back to *WinPC-NC*, and immediately see the result of the changes.

Show stop watch

Display the job stop watch. The clock starts when the job starts and stops when the job stops. The time is displayed in hours, minutes, and seconds of elapsed time.

Stop watch in WinPC-NC

Knowing the exact length of job is often useful. For instance, it is a critical piece of data necessary for costing commercial work.

Display machine position

In *WinPC-NC,* the display of the real-time machine position data can be enabled/disabled. The data appears in a small window.

Display NC program window

When this feature is enabled, a window displaying the currently loaded NC program will appear. A *cursor bar* is updated in real-time to indicate the command currently being executed.

Postscript and DXF files cannot use this feature. The maximum number of lines that can displayed is 300000.

Using the side panel in standard view, the NC window is always integrated and displayed during job process. However, it is also possible to use the free NC window which is activated by this parameter.
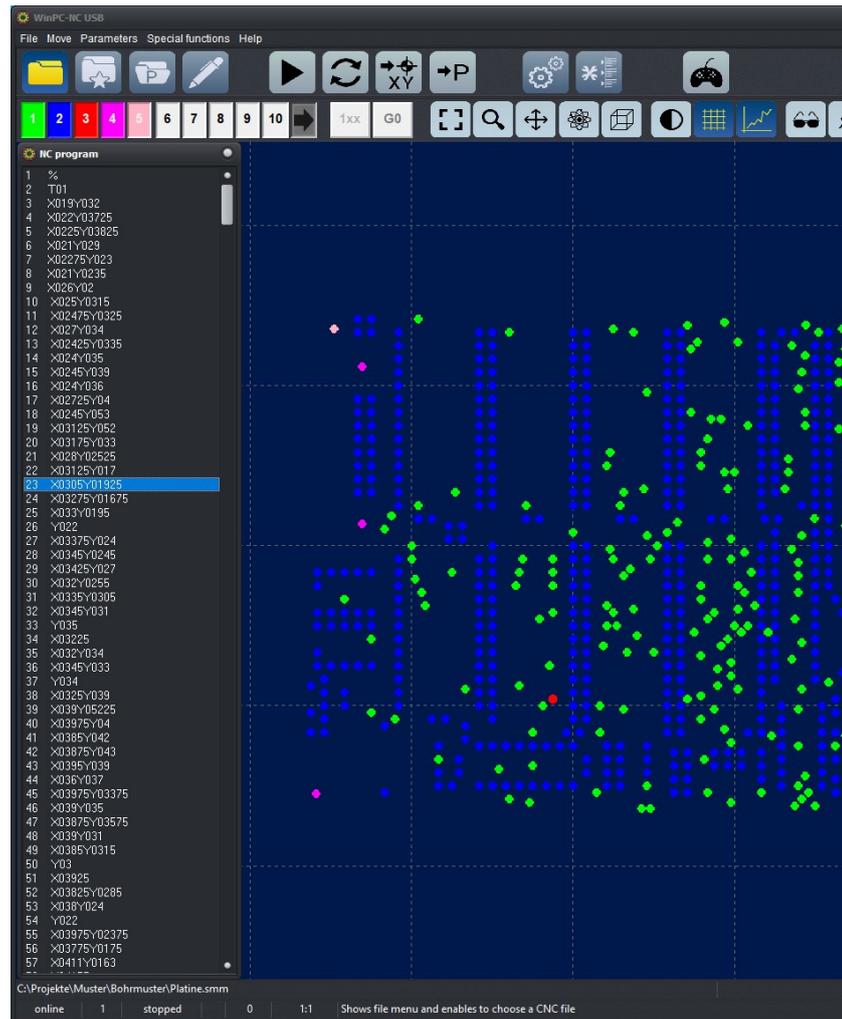
Program window during operation

Save last position

> *WinPC-NC* can save the actual machine position after each movement and job. This is useful when there are no reference switches available or homing is not always possible. The saved values remain valid even after *Exit* and *re-start*.

Exit by safety prompt

> If this parameter is set, a dialog appears asking whether you are sure you want to exit.

Travel button with safety prompt

> If this parameter is set, a dialog appears after pressing the travel button of the main screen (park position, move to zero,homing) asking whether you are sure you want to start.

Always save settings with exit

If this parameter is set, the **WinPC-NC** parameters are automatically saved with exit.

3D color shading

In **WinPC-NC,** the default view after loading a 3D file or relief is looking down onto the data. This can be visually confusing.
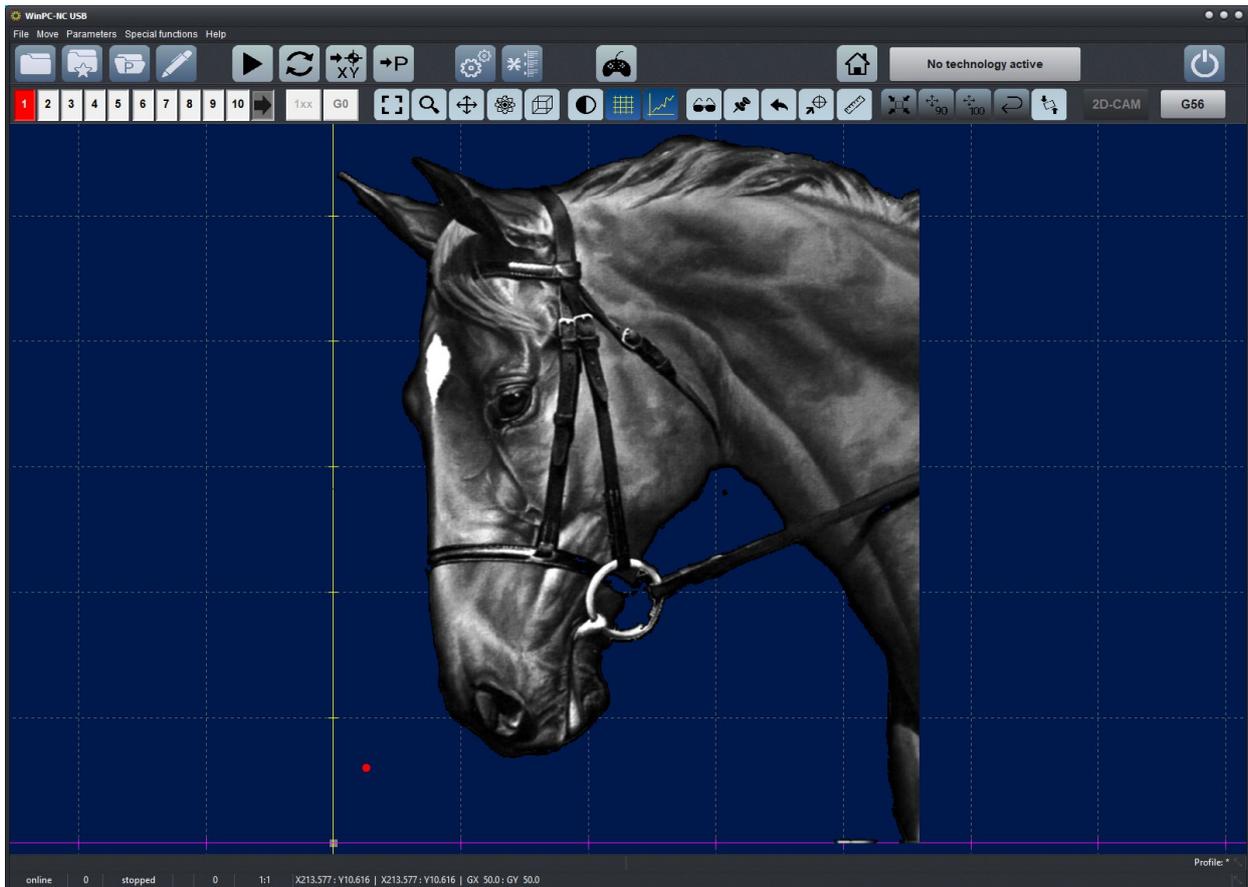
Color shade according for relief- for laser engraving or grinding applications

How the data is shaded – presented – can be selected. Choose one of the 5 possible methods that best suits you:

none,
2D data only   Here a 2D display is presented with lines and drillings in the selected tool color

From Z coordinates / from Z coordinates colored	Colors are shifted - lighter or darker - corresponding to the current Z height. The colored option will shift stronger as the regular one and this may cause considerable color changes.
From spindle speed / from inverted spindel speed	This option is often useful with grinding applications where different spindle speeds or PWM signal levels indicate the gray levels. Inverted shading a higher PWM value shifts the color towards lighter nuances.

5.5.2. Miscellaneous - Job process
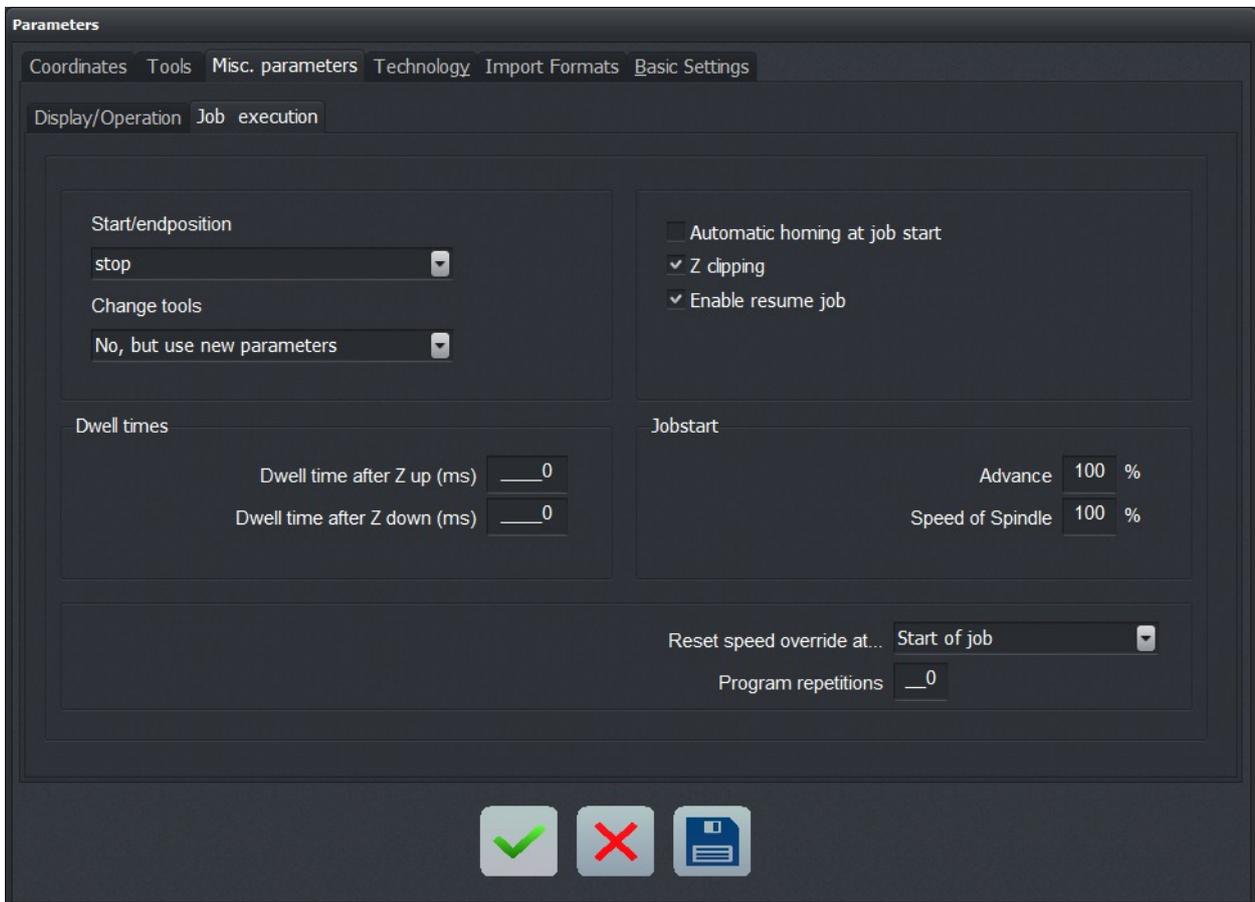
Parameter-Miscellaneous parameters-Job process

Start/end position

This specifies where the start and end point of each working process should be located. The machine also moves to the position after the initialization process.

There are 4 possible start and end positions:

Stop   **WinPC-NC** stops at the reference position after initialization and at the last coordinate at the end of each job.

Zero point   The machine moves to the zero point after initializtion and at the end of each job.

Park position   **WinPC-NC** moves to the park position after initialization and at the end of each job.

Zero point and clearance distance   The machine moves to the job zero point then lifts the Z-axis to the defined clearance distance.

Tool change

The parameter defines how **WinPC-NC** handles the tool change commands during a working process.

There are 5 possibilities for the tool change:

No   No tool change is performed, the entire working process takes place with the current tool.

Yes   Performs the tool change; remains stopped in the current position for every tool change.

Yes, at the park position   Performs the tool change; moves to the park position for every tool change.

No, but use new values   No tool change is performed, although new values for plunge depth, plunge speed, and feed speed are adopted.

Yes by ATC   Performs the tool change via an ATC.

Automatic homing at a job start

When enabled, a homing sequence is automatically performed by **WinPC-NC** prior to each job. This is **highly recommended** since rehoming will recalibrate the machine, eliminating any accumulated stepping errors.

Z-axis clipping

When enabled, Z-axis moves will be clipped. No Z-Axis move below the working area will occur.

Enable resume job

WinPC-NC, by default, does not allow an interrupted job to continue (resuming interrupted jobs often leads to errors). Care should be taken when enabling the ability to resume a job.

Prompt at job start of a previously stopped job - no Sidepanel
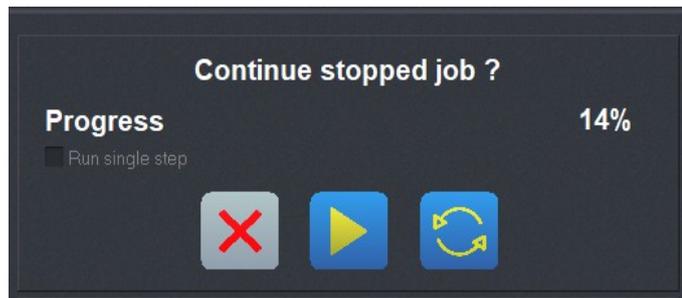
Prompt at job start of a previously stopped job - with Sidepanel

Dwell time after Z up and Z down

In many applications, it is a good idea (or a requirement) to pause briefly after the tool has been lowered or raised or before starting X-axis and Y-axis movements. For example, when working with flexible materials or when a tool takes time to come up to speed (or to stop).

The dwell time is specified in milliseconds.

Feedrate at job start
Spindle speed at job start

Here you can set feedrate and spindle speed override at job start.

Values from 5 % up to 200 % are possible and after starting the job speeds can individually be changed.

Reset override speed

The feed rate and spindle speed may be adjusted while a job is in progress. A change stays in effect until it is manually changed or one of the following occurs:

The following settings are valid :

Machine reset or program reset   Values are returned to their 100% values after a machine or program reset.

New file   Values are returned to their 100% values when a new NC file is loaded.

Job start   Values are returned to their 100% values when a job is started.

Program repetitions

The program repetitions parameter enables you to repeat a working process up to 999 times. Batch production is easily facilitated by this parameter and using the *Start* signal.

5.6. Technology - functions

The technology tab displays all of the available technologies (applications) that *WinPC-NC* supports. These technologies can be enabled or disabled, as necessary, according to your needs.

Special hardware is required by some of these technologies.

5.6.1. Technology - Job nesting
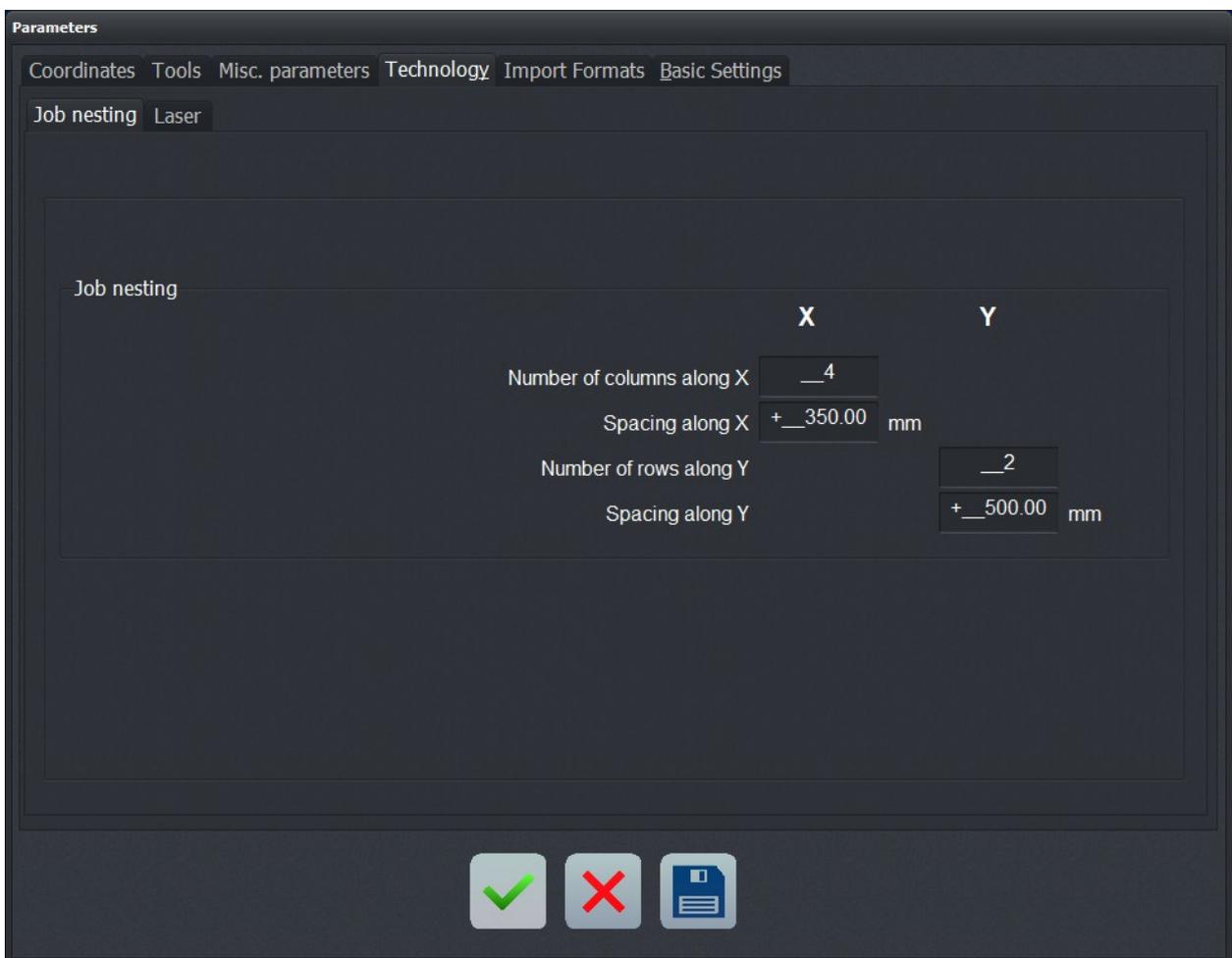
Parameter-Technology-Job nesting

Number of columns along X
Number of rows along Y

These parameters specify the numbers of pieces in each row and column. The product of these numbers is the total number of pieces to be produced.

Spacing along X
Spacing along Y

These parameters specify the distance, in millimeters, between the working pieces in the X and Y directions. These spacings must be large enough to prevent any tools from touching consectutive pieces.
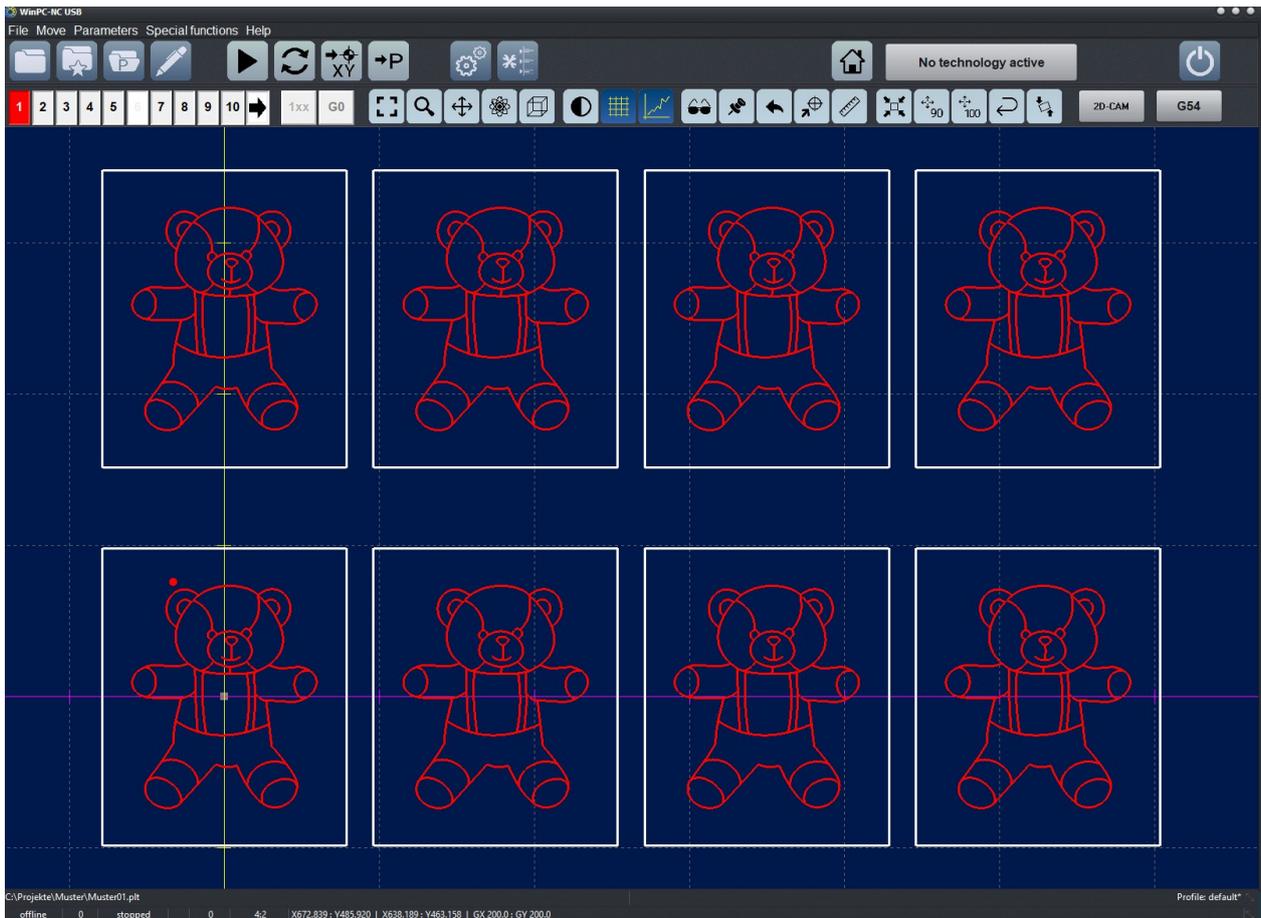
Nested job, 4 by 2 pieces

5.6.2. Technology - Cylindrical axis
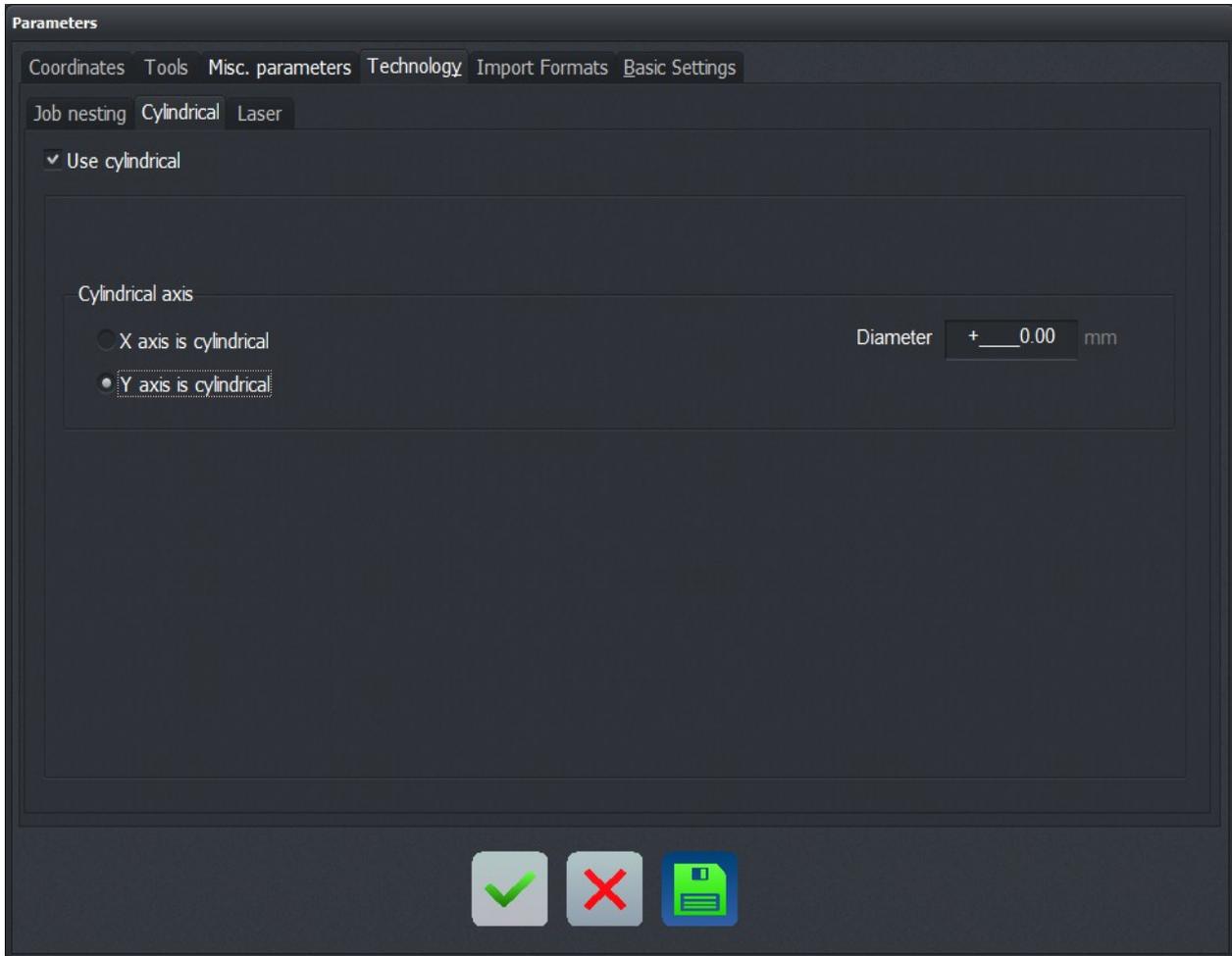
Parameter-Technologie-Rundachse

Use cylindrical

Enable or disable cylindrical engraving.

Cylindrical axis

You can define either X-axis or Y-axis as the cylindrical axis.

Diameter

The diameter of the cylindrical body to be engraved. is essential for calculation of the Y-coordinates and should be entered with greatest precision.

ℹ️ **Cylindrical engraving is possible only when using DXF- or HPGL- formatted NC data.**

Cylindric engraving example

5.6.3. Technology - Tangential cutting
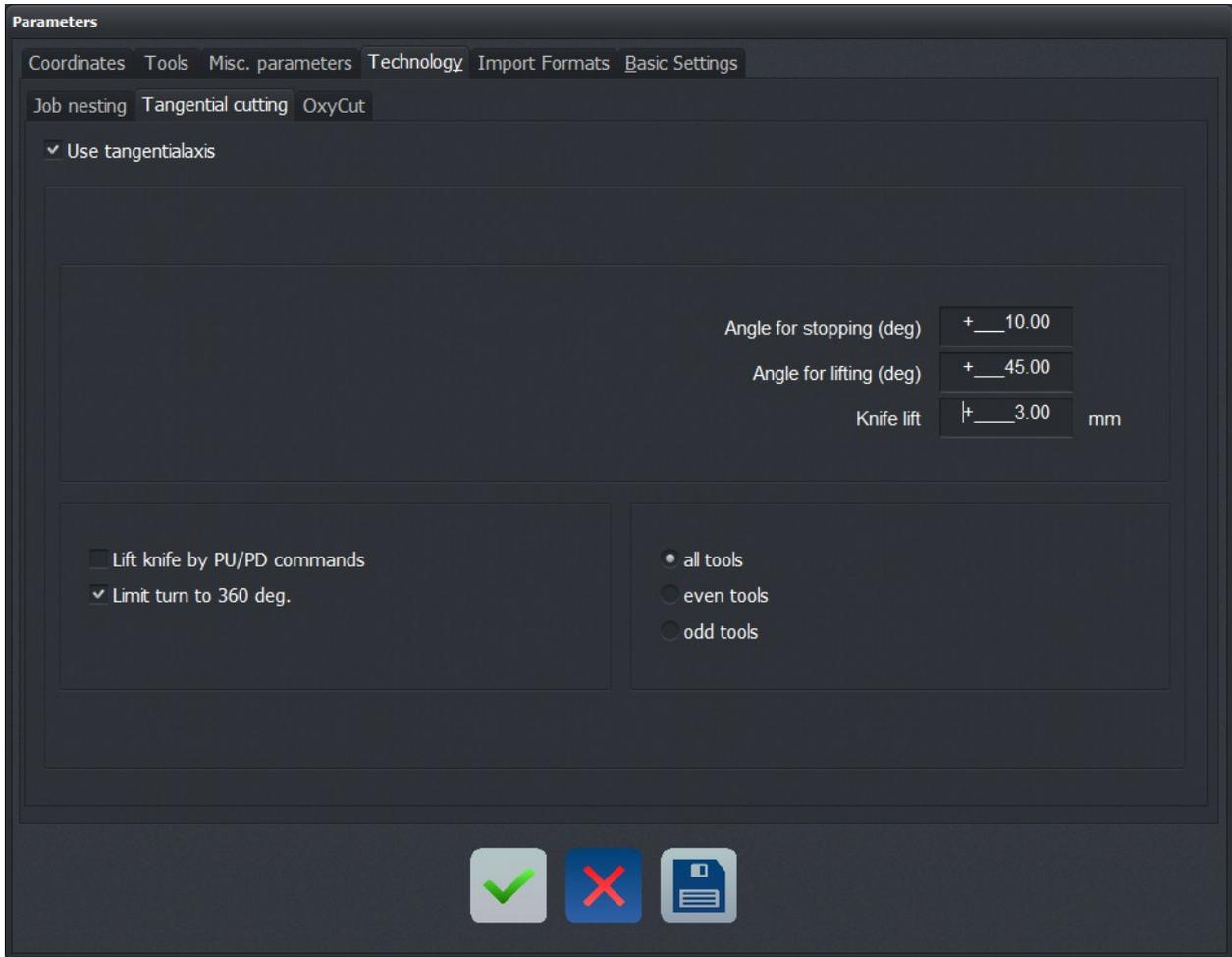
Parameter-Technology-Tangential cutting

Use tangential

Enable or disable tangential cutting.

Tangential cuts are only possible if the 4th axis is equipped with a reference switch or if a reference move has been carried out.
Otherwise the user is asked to execute a reference move.

Foil cutting with tangential cutter

Angle for stop/lift and lift height

When a rotation larger than the stop angle is requested, the cutter will stop and a re-set will be performed before the job continues.

No rotation of the cutter in the material

When a rotation larger than the *lift angle* is requested, the cutter will stop, the knife lifted, and a reset will be performed before the job continues.

The *knife lift* specifies the height, in millimeters, to which the knife must be lifted before a rotation. The height should be set such that no material damage occurs.

Limit turn to 360 degree

WinPC-NC limits the turn of tangenional axis to 360 degree. If a next move turns over 360 deg, movements stops, the knife lifts up, turns 360 deg back and continues cutting from 0 degrees onwards.

Selection of tools

The use of tangenional axis can be liited to certain tools used in the file. You can select all tools or even tools or odd tools. With unselected tools there is no control of the tangenional axis.

5.6.4. Technology - 3D-Printing

3D printing requires a special *print head*. The *print head* filament feed control must be connected to **WinPC-NC.** This requires using a 4th axis and connecting other inputs and outputs. Care must be taken to restrict speeds so as to not damage the *print head* or create a mess.
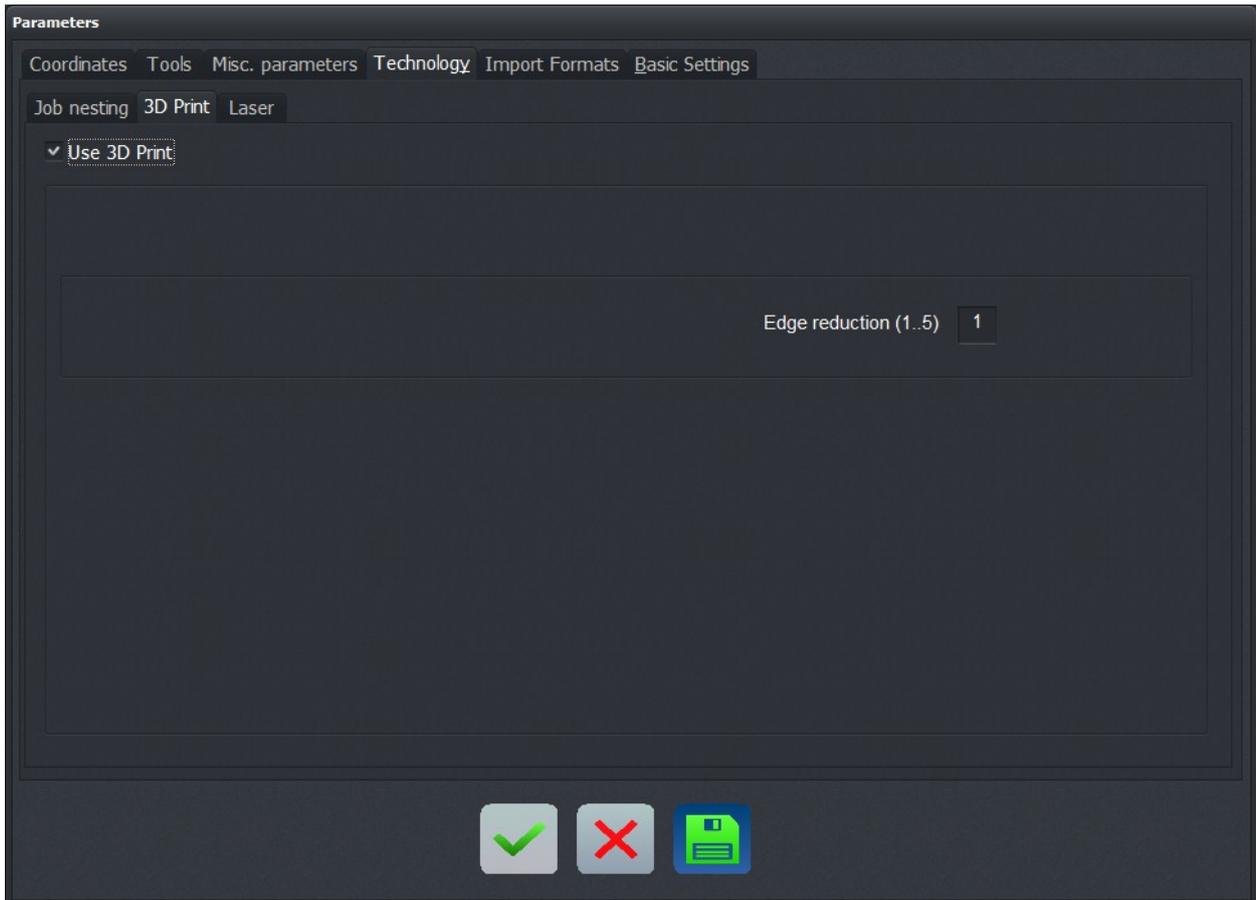
Parameter-Technologie-3D-prin

For detailed information on the exact handling and functions of the 3D printer with **WinPC-NC** please refer to our manual for print head.

Edge reduction

Specifies the edge reduction factor. The filament feed speed will be reduced step-by-step in corners and on edges. This can help avoid messy problems in these areas.
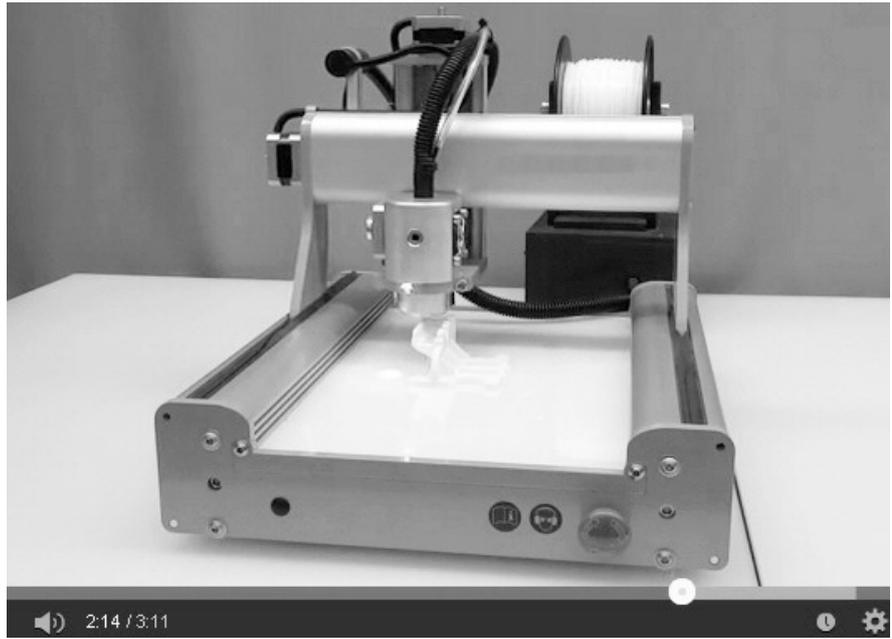
Machine with 3D-print head running a job

5.6.5.   Technology - Laser

For realizing any kind of engraving and cutting applications, *WinPC-NC* can also use a laser instead of a milling spindle. For setting the laser *ON/OFF,* we recommend to use output Q244, which always switches on for contouring.

If parameters of the technology function '*Laser' can* be displayed in the technology menu in *WinPC-NC*, the relevant function must already previously be set in *basic settings – equipment.*
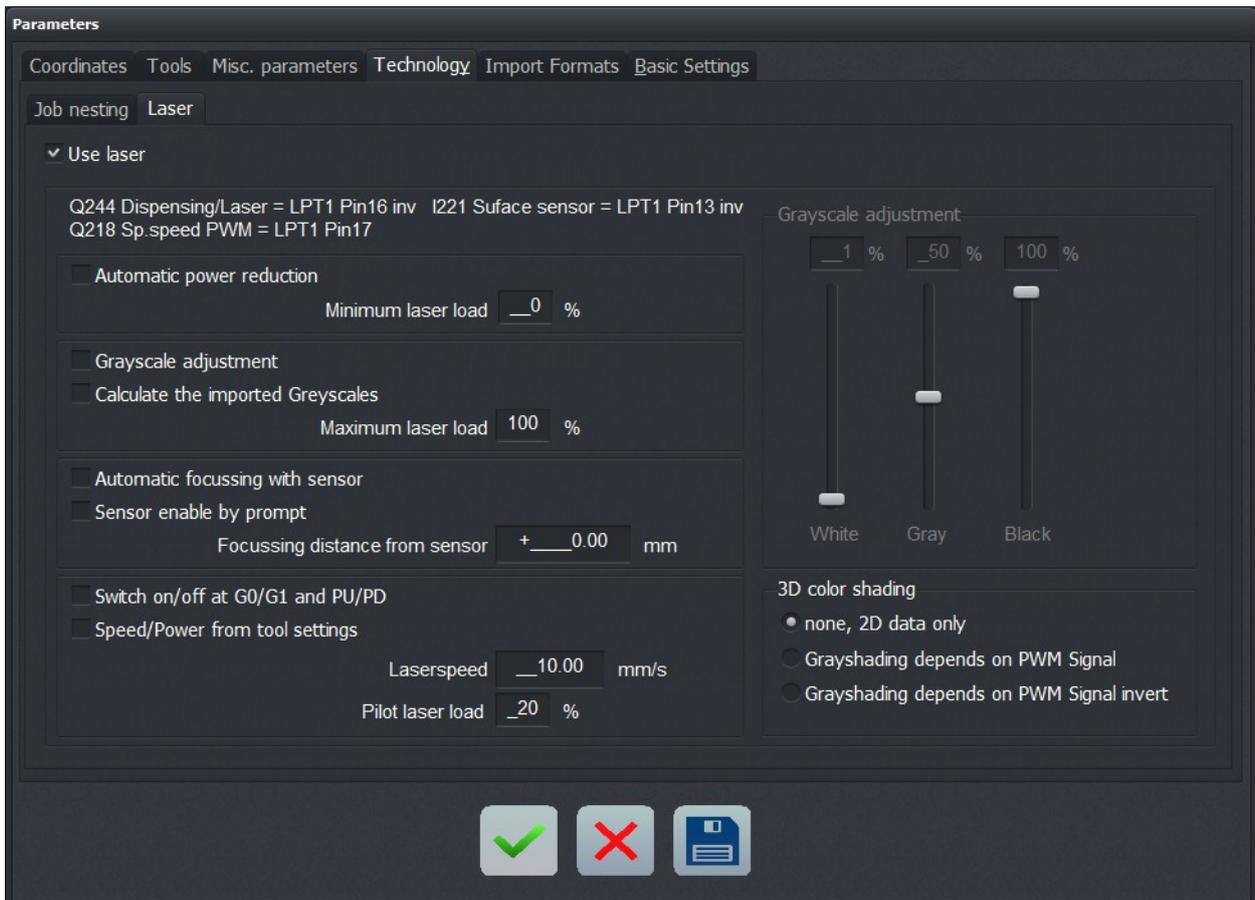
Parameter-Technology-Lase

Use laser

To use and activate the laser functions, check this box.

A more detailed instruction for controlling a laser can be found at our web site at www.Lewetz.de - at Help - Howto .

Q244, Q218, I221

The required signals to drive the laser are displayed here. To set up right pin assignment for the laser and its functions, refer to the quick start guide for installing the laser.

In generated NC data the laser is often switched ON with M10 and switched OFF with M11.

Win-PC NC **interprets these commands in the same way.**

Automatic power reduction and Minimum load

WinPC-NC performs an interpolation of the laser power depending on the current speed of your machine. The corresponding checkbox must be checked to activate this function. Laser power is automatically reduced during braking and slower driving to ensure that the work piece will not be burned*.*

automatic power control

The used material will determine if a reduction to 0% laser power is necessary and desirable. The minimum power can be entered here, which is used as minimum speed for the automatic power reduction. This is beneficial if the laser does not perform any visible marks on the working piece with 0%-20%. The minimum load should be set to 20%.

This reduction takes place additionally to the power values or spindle speeds in the loaded nc file.

Grayscale adjustment

By activating the grayscale adjustment function the grayscale input field on the right hand side of the parameter window will be displayed. The values entered here will be considered for the laser power calculation in WinPC-NC. The grayscale function allows further compensation of the nonlinearity of the laser diode to achieve better results.

Engraving of gray scale areas

Values for the grayscale adjustment can be entered on the right hand side of the parameter field. The values can be set from a previous run or can be determined with the laser test function.

To find out how to adjust the gray scale level, refer to the quick start guide for the laster test functions.

Scaling of the imported Grayscale / Calculate the imported Grayscales

The values to control the laser will be between 0%-100% on default *WinPC-NC* uses the spindle speed information in the nc file if the parameter has been deactivated. However, only values ranging from 0-100 are considered during the job. Any value larger then 100 is automatically equated to 100% laser power. Since this setting is not very ideal, *WinPC-NC* performs automatic scaling when loading the nc file if this parameter is active. Nc files containing spindle speeds between 0-100 are considered as such and no scaling takes place.

Job files with spindle speeds between 0-255 with at least one speed value greater than 100 will be scaled to 0%-100% automatically.

Job files that contain spindle speeds greater than 255 are automatically scaled that the highest contained speed value is equated with 100% and the remaining values be interpolated.

When creating your job files, be sure to set the spindle speed values between 0-100 and this parameter is not relevant for you. Furthermore, this parameter is enabled by default and should only be disabled if automatic scaling is not desired.

Maximum load

This parameter specifies the maximum laser output power in percentage in which the file is processed. It must have a value between 0%-100%. This is specially designed for materials that needs a very small laser intensity or for lasers with a high laser output power. The value entered here is automatically included in the calculation of the gray scale adjustment.

The laser power now becomes linear between the minimum power and the maximum power interpolated. This happens depending on the spindle speed values specified on the nc file. Additionally the current machine speed can be considered for the power calculation with the previous described parameter automatic power reduction.

Automatic focussing with sensor and Focussing distance from sensor

This checkbox activates the automatic Z zero point finding function. This can either be done through a surface sensor placed on the work piece or a sensor used on the laser. After *WinPC-NC* detects the activation of the sensor, it automatically reverses the value of the parameter Focussing distance from sensor. This va-

lue must be set to match the height of the focus point of the laser. If a flying surface sensor is placed on the work piece to determine the zero point, the height of the sensor can be deducted from the value of the parameter Focussing distance from sensor.

Sensor enable by prompt

This parameter must be activated, if the surface sensor or sensor for automatic Z zero finding must be placed on the work piece before the measurement. Now *WinPC-NC* waits before the start of the measurement until the user acknowledges the positioning of the sensor and before the job start until the user acknowledge that the sensor is removed.

Switch on/off at G0/G1 and PU/PD

Selecting this parameter enables or disables the automatic control of the laser through *WinPC-NC*. This parameter should be set if the nc file does not include the correct commands that will turn off the laser during empty runs.

In GCode files, the output for the laser is switched on with M10 command and switched off with M11 command. If this parameter is activated, switching the laser on and off are based on the G-commands. In this case, the laser is off at G0 moves (empty runs) and on at G1, G2 and Gx moves.

For HPGL files the pen-up and pen-down commands are used to control the laser. For DXF files, the empty runs are used to switch the laser on and off.

Enabling this parameter prevents errors that may arise with using certain postprocessors.

This parameter should be activated if the laser is switched on during empty runs. Activating the parameter is suitable for troubleshooting as compare to using other outputs e.g. *Q242 Spindle* or *Q243 Cooling*.

Speed/Power from tool settings

If that checkbox is activated WinPC-NC does not use the laser speed and power values from the technology page but takes the value from the Tools parameters. You can enter a value for each tool there and can select the desired values by using the corresponding tool in the nc file.

Laserspeed

The parameter Laserspeed can either be entered directly or be determined during the speed test. How to find the right laser speed for the used material with the *WinPC-NC* laser speed test function is explained in the quick start guide for the laser test functions.

The value entered here corresponds to the maximum speed used to calculate the speed dependent power reduction and to the maximum speed used during the working process.

Pilot laser load

The mounted laser can easily be used as a pilot laser with *WinPC-NC* to measure the work piece X-, Y- zero point before starting the job. The desired percentage value for the pilot laser must be entered here.

The pilot laser is only used as a marker to align the machine and must be kept as low as possible. It must be adjusted in that way, that activation of the pilot laser does not affect the underlying material or anything else.

5.6.6.   Technology - Grinding

Grinding with control of the grinding pressure is available. For example, grave stones or other stone surfaces can be inscribed and/or images can be embossed.
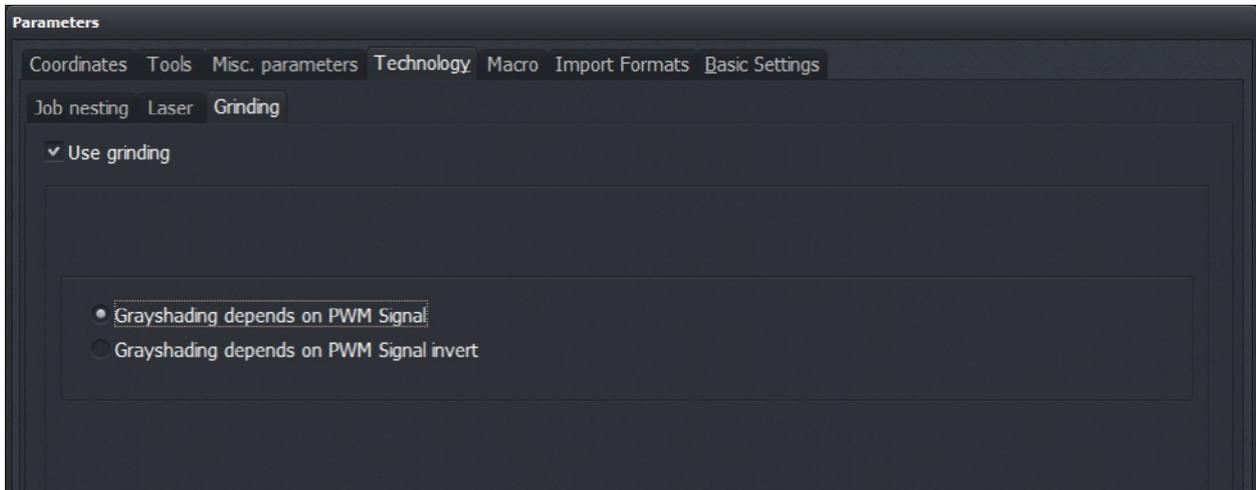
Parameter-Technology-Grinding

Grey scale depends on PWM signal
Grey scale depends on PWM signal inverted

Images can be embossed as grey scale data. The grey scale determines the spindle speed.
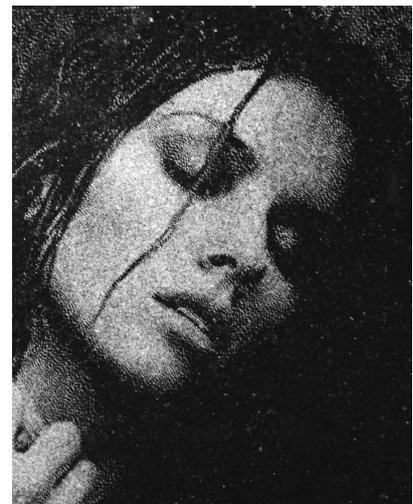
5.6.7. Technology - Oxy and plasma cutting

Due to an integrated technology function for plasma and flame cutting **WinPC-NC Professional** is able to handle different control units by simultaneously managing the height compensation.

Example - plasma cutting

All relevant settings can be made in the dialog *Technology – Plasma*.

The technology function 'Plasma' is only available in *WinPC-NC Professional.*
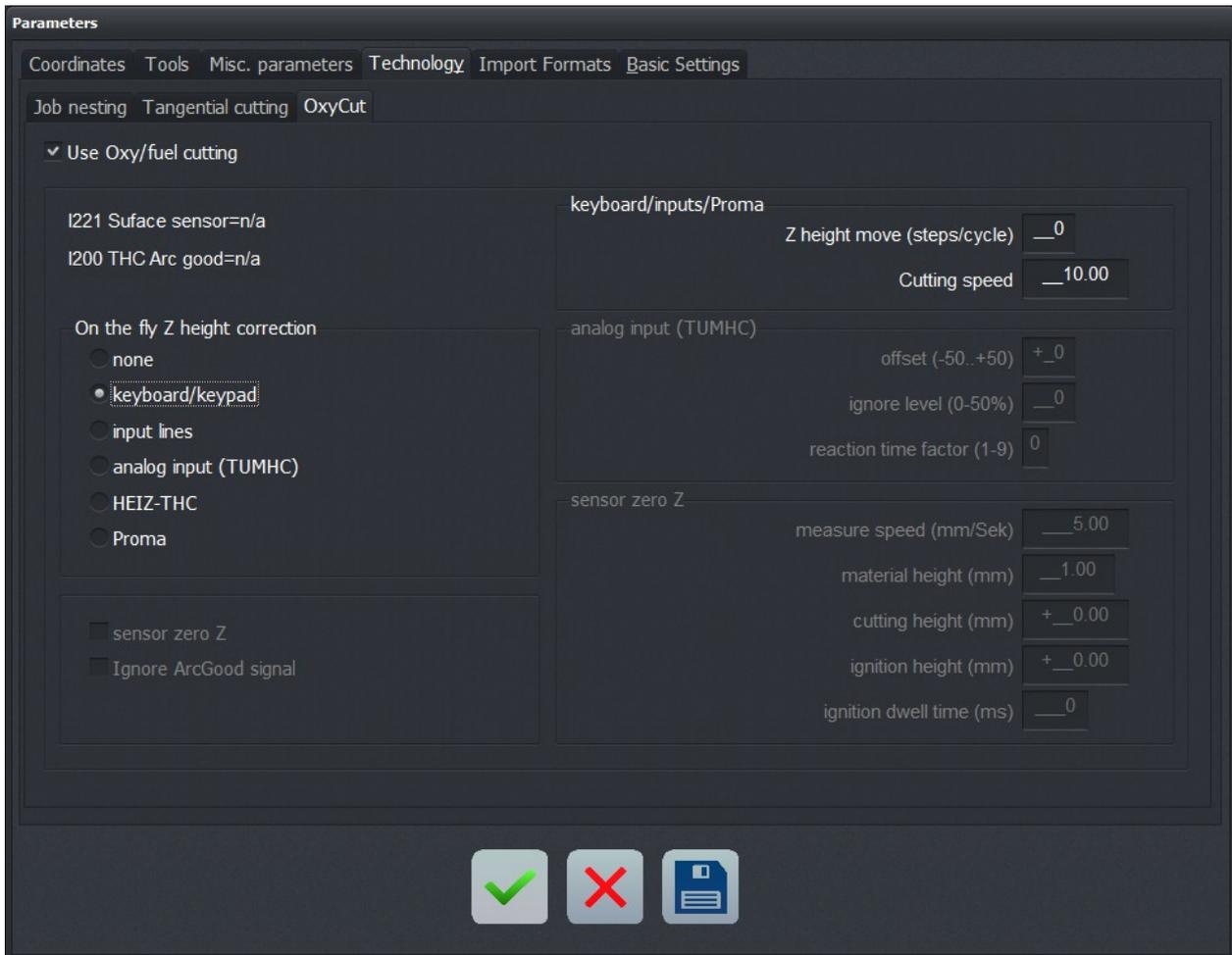
Parameter-Technology-Oxy/plasma cutting

Use Oxy/plasma cutting

As it applies to any other technology functions this one can be set ON/OFF as required.

Mode of operation

With activated function ,Z-correction' **WinPC-NC** monitors inputs and buttons for correction during the job process. If a correction command is identified **WinPC-NC** moves the Z-axis immediately in cyclic mode by the set step size upwards or downwards. Correction is finished and Z-movement is stopped either by released button or input signal is LOW again.

Correction of the Z-axis is only active, if the tool is lowered during the job or if it moves in or along the material. This applies for all movements which are executed by PD (PenDown) commands in HPGL files or G1/G2/G3 commands in DIN/ISO programs. There is no compensation with empty runs and raised tool.
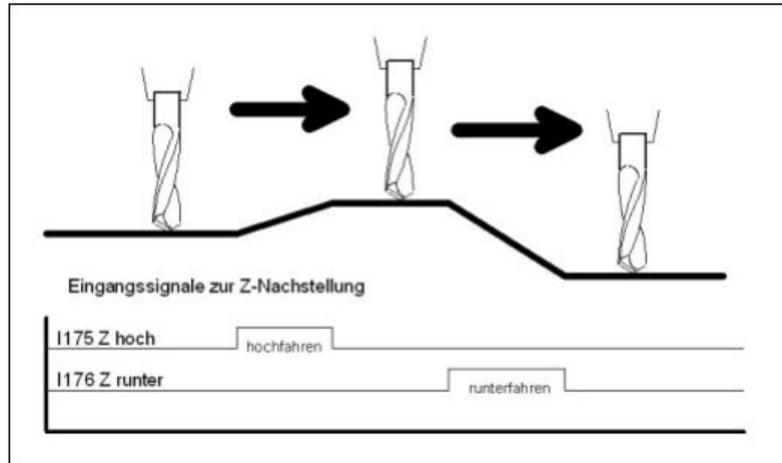
Z height correction on non flat material

required input/output signals

The below described options for height compensation require following inputs and outputs.

I199 THC Error   optionally, from control unit
I176 THC Z move down   from control unit
I175 THC Z move up   from control unit
I221 Probe/Sensor   surface sensor find part zero
I200 ArcGood   from control unit, electric arc
Q116 THC Starting   from control unit
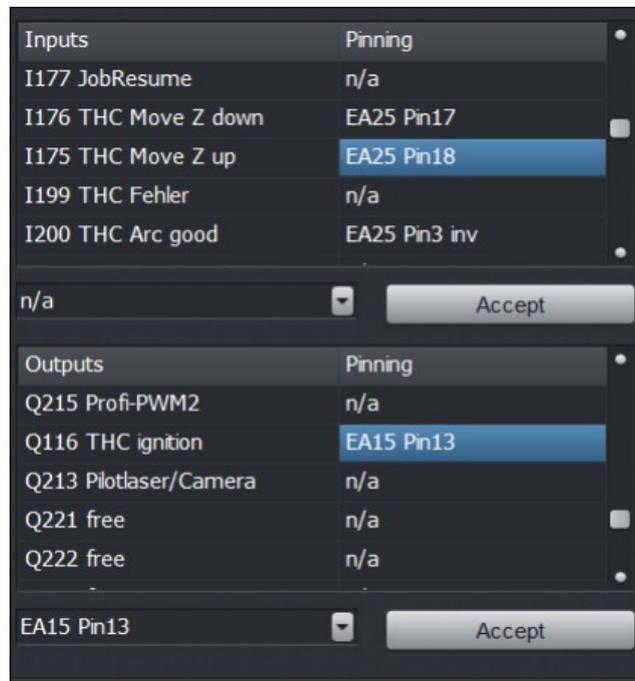
These pre-defined inputs and outputs are automatically operated when a cutting job is started and the process plunging and flame cutting is executed and controlled. However, inputs and outputs must be pre-defined in the signal assistant and it is not done automatically.

Z height correction

Defines mode of height correction of the Z axis with non-planar plates. Various options are selectable.

OFF

No height correction or compensation with external controller, e. g. as provided from Micompan.

Keyboard or Keypad

Height compensation is released in interactive mode and during cutting process with the keys (PLUS) and (MINUS) of the keyboard or by keypad. By each pressing of keys, the number of steps of the parameter *height compensation* (steps/cycle) are proceeded and thus it means correction of the Z-axis.

Analog input TUMHC

Compensation via analog input is only possible by using a specially conceived hardware component which measures directly the arc voltage and provides a proportional analog value. This value is directly processed and analyzed by the axis controller CPU. The result can be optionally changed by settings OFFSET, DEADZONE and REINFORCEMENT.

This is a customized technical development and not available for other users.

HEIZ-THC and Proma

The setting supports THC controllers, which are ready for use of following companies: CNC-Step, Proma and Hyperther

Connection can also be made via an optional LPT2 port on the axis controller and it is compatible to Mach3.

Find Z zero with sensor

Find Z zero

It is even possible to activate a function for *find part zero* for all modes of height compensation, provided that a suitable sensor is available with each groove into the material, height determination and burn-in process are made automatically.

Mode of operation

Some parameters are used to *find Z zero* and which are interpreted as follows:

1. The machine moves to the new groove point in XY position on the Z level of the clearance distance.

2. The Z axis moves down according to the measuring velocity until the sensor (I221 button) displays contact with the material and then stops in the position.

3. The Z axis moves up to the defined height of grooving, starts burner (Q116 THC start) and waits for 200 ms.

4. The Z axis moves down to the defined cut height and requires for this move exactly the time indicated for plunge delay.

5. In this position **WinPC-NC** waits for signal *Arc good* (I200 Arc good) of the THC controller and recognizes that the material has been successfully cut.

6. Then contouring starts and signals for permanent height correction (I175 and I176) are analyzed after a dead time of 200ms.

Cutting speed

The cut is made by the defined speed after plunging into the plate. A good result concerning cut and edge is extremely dependent on this setting and should always carefully be selected. The same applies to the material thickness.

5.6.8.   Technology - Dispensing

Specific dosing functions

WinPC-NC Professional provides a technology function for dosing. This function is only active with a defined dosing output on the axis controller and the output **Q244 Dosing** is available.

The technology function *Dosing/dispensing* is only available in *WinPC-NC Professional.*
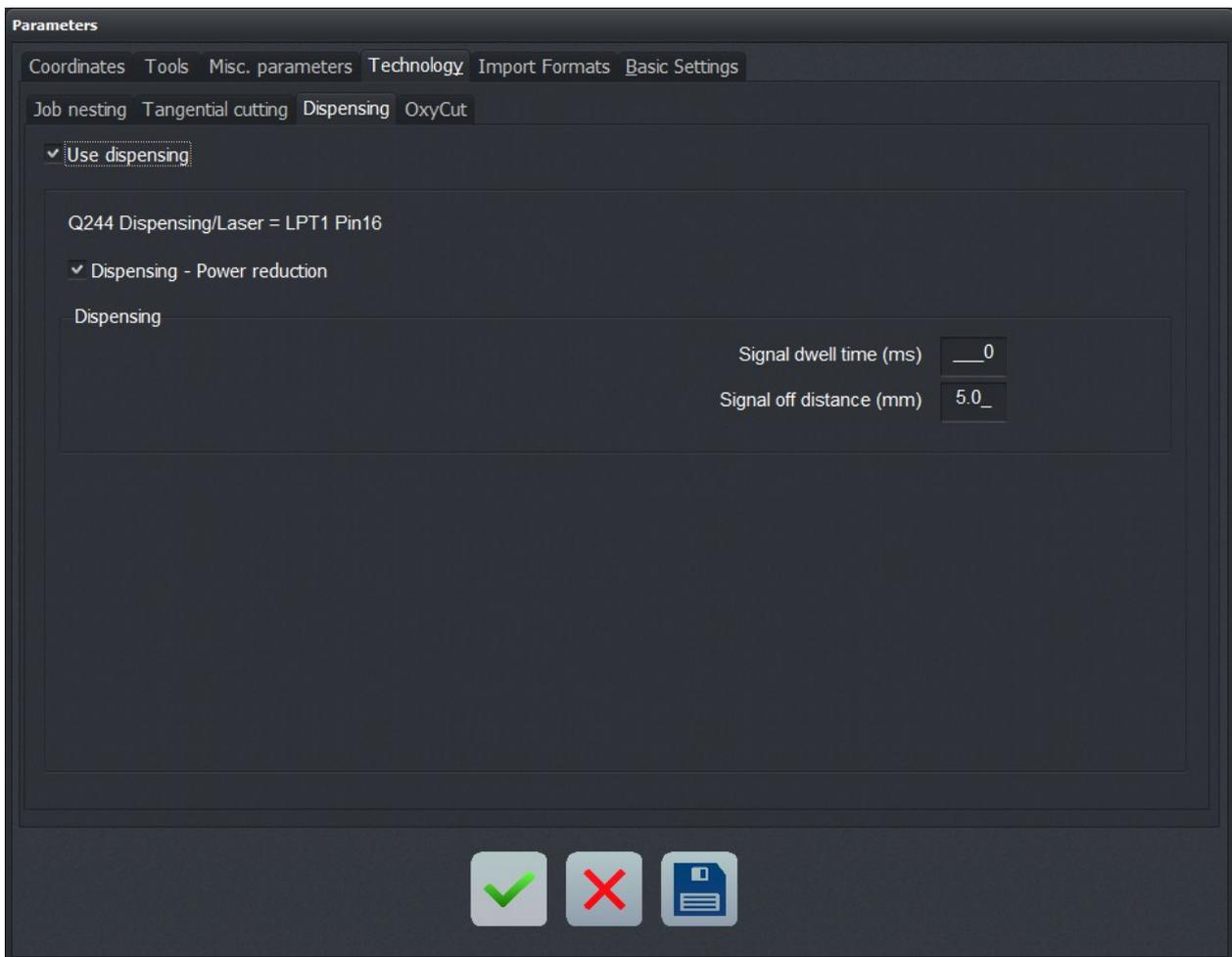
Parameter Technology dispensing

Output set-up time

The set-up time defines a dwell time which starts after activating the dosing signal and prior to movement. Thus it is possible to build-up a specific dosing pressure.

Output follow-on distance

The follow-on distance defines a distance whose dosing output is already set inactive prior to the end of the contour. This procedure avoids further dripping of the dispensing material.

Dosing power reduction

With this function the PWM signal is used for controlling – even speed-dependent - the dosing pressure. The value and thus the pressure are automatically reduced with decreased feet rate in tight radii and at edges in order to avoid excessive dosing.

Dosing applications with/without defined overtravel distance

5.7. Macros

WinPC-NC has a powerful, user-definable, macro facility de-
signed to enable or enhance a wide range of work flows and ap-
plications. Custom hardware requirements are easily handled via
this facility.

Macros must be enabled before they can be used in
***WinPC-NC*.**
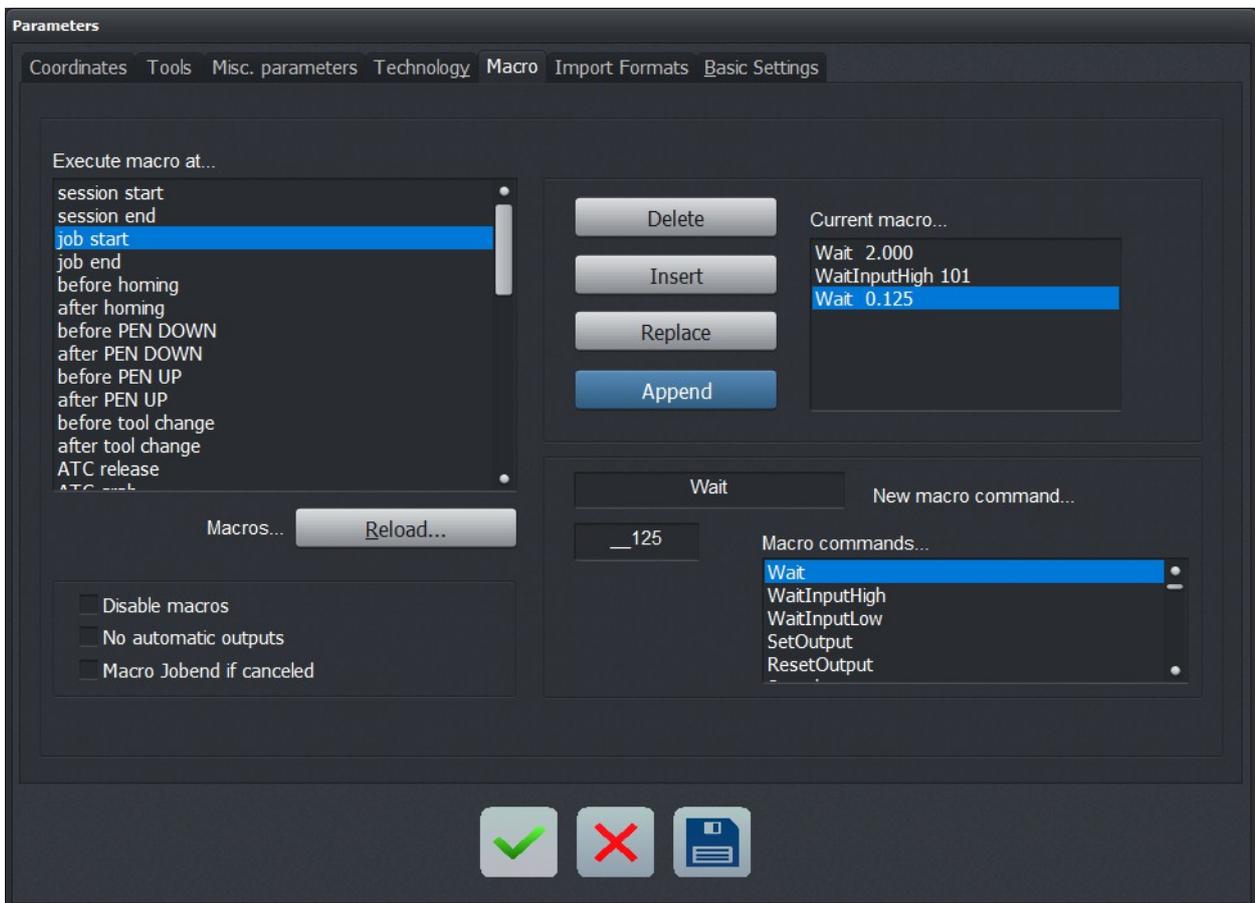
Parameter-Macro

Flexible with   *WinPC-NC* always stores its macro definitions in the
macros   WINPCNC.MAC file when the parameters are saved. They are
                       always valid and apply to all projects.

The *Reload* button will load most recently saved macros from the WINPCNC.MAC file.

The dialog box for macro handling is divided into several parts:

left window   The list of *WinPC-NC* events where a macro can be used.

Window top right   The current macro. A new macro is assembled in this box.

Window down right   The list of all possible macro commands suitable for the current macro.

Editing boxes   The selected macro commands are combined with the necessary parameter values here (e.g. dwell time after a move).

Editing buttons   The current macro command may be deleted, replaced, or edited. A new command can be created.

A macro must be assigned to a specific *WinPC-NC* event (e.g. job start, after a tool change, when a special tool is selected). It will be executed when that event occurs during a job.

Some of the macros are only available in specific contexts (e.g. G-code or HPGL NC files). For example, the macro1 macro can be executed when the M90 M-command is found in a G-code NC file. Further information about the specific context macro capabilities is provided in later chapters.

Create macros   **Macros manipulation procedure:**

- Select an event by double-clicking in the **Execute macro at …** box. This is the event with which the macro is to be associated. The **Current macro…** box will display "No macro defined" for a new macro or display the commands of an existing macro.

- Edit the selected macro by double-clicking the lines you want to change. The selected command will appear in the editing box. Once the command appears in the editing box, values can be assigned. Click *Replace* or *Insert*, as necessary, to change the selected macro.

- To create new macros or macro commands select the command from the list in lower right menu, enter the needed va-

lues and click on *Insert* or *Append* button to move command into macro list.

- To delete a macro command, move the bar to the corresponding command and click the *Delete* button.

Macro commands

Most of the macro commands are self-explanatory. A few of the more unusual ones are listed here:

Wait time	Wait time is in ms
Wait InputHigh *num*	Waits for HIGH to appear of input '*num*'
Wait InputLow *num*	Waits for LOW to appear of input '*num*'
Set OutputHigh *num*	Sets output '*num*' to HIGH
Set OutputLow *num*	Sets output '*num*' to LOW
Speed spd	Speed setting for future moves in mm/secs.
Move *xxx*	Moves a relative distance
MoveAbs *xxx*	Moves to absolute position xxx (i.e. in machine coordinates)
Spindle ON/OFF	Spindle is turned on/offl
Cooling ON/OFF	Cooling is turned on/off
Dispensing ON/OFF	and more output signals
Job running ON/OFF	
Job finished ON/OFF	
Molette open/close	Opens/closes molette
Wait Spindle Stop	Waits for HIGH to appear on the 'SpindleStop'. Only available with certain spindles that are wired appropriately.
WaitStart	Waits for HIGH to appear on the Start signal (input I255).
HomingX	Initialises axis to its switch
OffsetX	Enables an offset to all following axis coordinates, useful when switching between Z axis heads (***WinPC-NC Professional*** only)

Zero Z probe	Moves Z axis to probe and saves new zero point, used at oxy/plasma cutting **(*WinPC-NC* Professional** only)
Spindle speed	Sets new spindle speed
Offset XYZ prompt	Similar to single offset command but opens entry fields for offsets to all axes. Offsets can be defined interactively. **(*WinPC-NC* Professional** only)

Inputs and outputs have formal names (e.g. I100, Q115). Do not use an LPT port pin number. The formal inputs and outputs must be associated with specific hardware ports and pins before they can be used. Use the Signal Wizard to create the necessary associations.

Example 1
Release tool

Macro releasing a tool from a changer magazine:

Speed 80,00	Set the speed to 80 mm/s
MoveZ 156,34	Move down with the Z-axis
Speed t 5,00	Set speed slower
Wait Spindle Stop	Wait until spindle has stopped
MoveY 10,00	Move the tool into the chuck holder
MoletteOpen	Open the molette
Wait 500	Wait 500 ms for pneumatic system
Speed 2,00	Set very slow speed
MoveZ -5,00	Move 5 mm up from tool
Speed 80,00	Set speed back to fast
MoveZ -151,34	Move Z-axis all the way up
MoveY -10,00	Move Y-axis back to starting position

Example 2
Insertion when
flame cutting

Macro defining an insertion procedure when flame cutting. It is always performed before *PenDown* commands.

MoveZ 50,00	Move down with the Z-axis
Wait 10000	Wait 10 seconds at pre-heating position 1
MoveZ 20,00	Move down again by 20 mm

Wait 2000	Wait 2 seconds and pre-heat
SetOutputHigh 100	Switch on oxygen with output 100
Wait 500	Wait another 500 ms before movement

Please note that specific macro commands as for example *Offset* or macros concerning special functions, for example to find part zero, are solely available in *WinPC-NC Professional.*

Signals only via macros

output signal automatically or by macro control

By default, **WinPC-NC** generates certain output signals as it runs (e.g. 'Job running' while a job is running, spindle control signals, coolant pump on/off).

Clicking this checkbox disables the automatic output signal generation. Output signals must be generated explicitly from within a macro. This makes it possible to control, explicitely, every machine operation.

Disable macros

This checkbox disables macro and runs functions and jobs without macros.

Macro job end at canceled job

This setting performs the *job end* macro even if a job is canceled and not finished correctly.

5.8. Import Formats

WinPC-NC understands many NC data formats. These data can be displayed and used for controlling a machine.
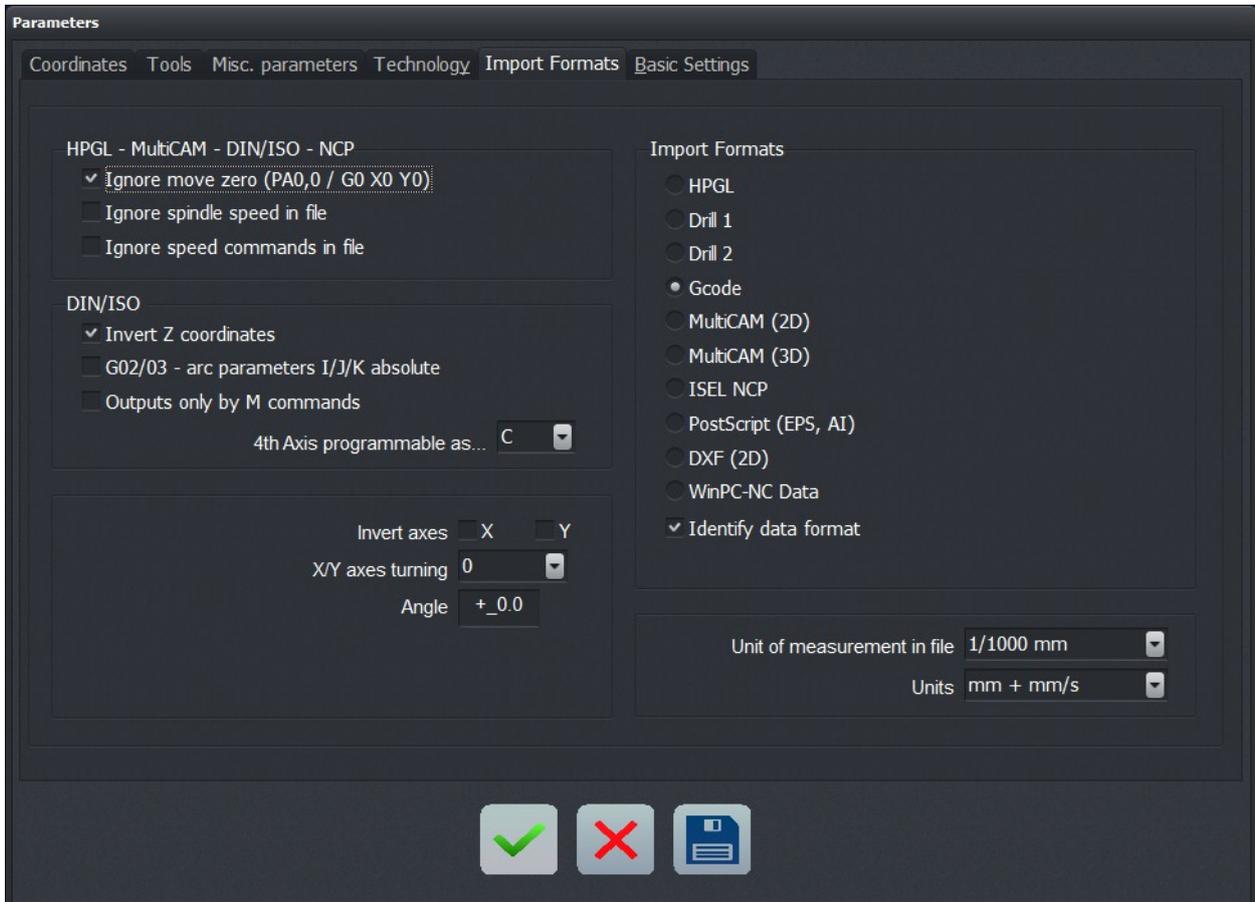
Parameter-Import Formats

WinPC-NC understands various NC data formats and is able to import these files, display them and perform the movements on the machine. At present, the commonly used plot format HPGL, MultiCAM, two drilling formats, DIN/ISO, DXF(2D) and ISEL NCP are possible. Furthermore *WinPC-NC i*s able to recognize vector information of postscript EPS and AI files, e. g. as created by various Adobe programs.

HPGL is from the wellknown plotters made by Hewlett Packard and is supported by almost every CAD or drawing program. The MultiCAM format as well as the Extended HPGL format is commonly used in the USA and provides specific 3D expansions to HPGL commands.

Plot and drilling formats

The drilling data interpreters recognize data in the following standards: Sieb&Maier1000, Sieb&Maier3000 and Excelon. i. e. processing job files with these formats should be done without any problems.

Generally NC machines are programmed with the G-code of the DIN/ISO format.

Detailed information on commands and description of the individual formats are enclosed.

Concerning faults and disturbances which may occur during the operational process, e. g. unknown or non-relatable commands, the job file can be analyized and post-edited by using this guide.

WinPC-NC usually identifies the loaded file format automatically and loads according import filters.

Automatic identification of the format

By default, **WinPC-NC** will attempt to determine the type of NC data as it is loaded (automatic recognition). Rarely, this identification process can fail. If this occurs, explicitly select the NC format of the data to be loaded and try loading the data again.

Ignore movement to zero (PA0,0)

In HPGL and MultiCAM files a PA0,0 command is often found at the end of the program. Checking this checkbox will suppress interpreting this command.

Ignore feed rate and spindle speeds

HPGL, MultiCAM, and G-code formats may contain commands to set the spindle speed and feed rates. Checking this checkbox will cause any NC speed control information to be ignored; tool data will then be taken from the **WinPC-NC** tools parameters.

Invert Z-axis

By default, **WinPC-NC** interprets negative Z-axis positions as *up*. Most CAM programs interpret negative Z-axis positions as *down* (plunge). Checking this checkbox will invert the default **WinPC-NC** behavior.

G02/G03 - I/J/K absolute

There are several dialects of G-code. In one, the G03/G04 (arc) parameters I/J/K are specified as absolute coordinates, while in the other the I/J/K are specified as distances relative to the current position. This parameter enables you to distinguish between the two.

Output signals with M commands

In G-code programs, external control is handled via "M" commands (e.g. spindle, coolant, tool changes). When this checkbox is checked, **WinPC-NC** will perform the actions associated with M commands as they are encountered. Otherwise the actions will occur at the beginning of the job.

4th Axis programmable as...

Letter for programming
The 4th axis is addressed in G-code programs via a letter. Axes parallel to X, Y, and Z are designated, respectively, U, V, and W. Rotations around the X-, Y-, and Z- axes are designated, respectively, A, B and C.

Mirror/Invert axes

The X- and Y-axes and their coordinates can be independently inverted. The change is immediately visible in the graphical display.

Rotation of X-/Y- axes

The NC data can be rotated around the zero point for a better stock placement. Rotation is in 90 degree steps.

Angle

A file can be turned by the set value with this parameter.

This setting is used for precise aligning of objects which are not clamped parallel to the axis and can be solely defined from -20 up to +20 degree.

Unit of measurement

Diverse pre-defined measurements
The unit of measurement for working data must be defined using this parameter. All coordinate values in the working file are related to a particular dimension.

Units may be millimeters or inches. HPGL files are usually in units of 1/40 mm or 1 mil; drilling data are usually in 1/100mm or 1 mil.

The units for HPGL, DXF, EPS, and AI files are automatically set when the file is loaded.

Units in the NC program

Specifies how NC file distances and speeds are understood by *WinPC-NC*.

There are three different settings :

- Millimeters and millimeters/second (mm and mm/s)

- Millimeters and millimeters/minute (mm and mm/min)

- Inches and inches/minute (inch and inches/min)

Please note ! Units must defined same in CAM program or in loaded ncfiles and in *WinPC-NC.*

Especially speed units are very important and it causes wrong feedrates in job if speed in file are mm/min and *WinPC-NC* interprets it as mm/sec.

---

6. Setting up a New Machine

Adaptation to the machine

Once the **WinPC-NC** software has been installed, a new machine must be set-up and tested before it can be used for production work. The set-up procedure generally only has to be done once.

If your machine came bundled with **WinPC-NC,** the set-up procedure can be skipped. Follow the instructions that came with the machine to load the parameters appropriate to your machine.

ncUSB box with two LPT-compatible ports

The additional module ncUSB provides two LPT like ports and pin compatible interfaces. For reasons of simplification, in this document we continue to use the term LPT port.

On delivery, **WinPC-NC Professional** comprises the axis controller CPU CNCCON. Different types of constructions are available, either as pure CPU card for being integrated in control units or as standard casing with external power supply unit.

Axes controller in standard housing, **CNCCON-S**

However, it is also possible to use our **CNCMAX** control units comprising, besides the CPU, amplifiers for 3-4 step motor axes.

Controller **CNCMAX** with axes logic and stepper drivers included

WinPC-NC Professional *optional with opto-coupled signals*

Optionally each axis controller CPU can be equipped by an expansion card providing control cabinet compatible 24V signals and 24 opto coupler inputs, 8 opto coupler outputs and an analog output 0-10 V.

6.1. Connect ncUSB or axes controller to your machine

After software installation and connecting the additional module ncUSB or the machine to the computer, automatic hardware recognition should be started and, after a short moment, a message is displayed with the information that the new hardware is ready for use.

Warning !

For energy saving reasons all Windows operation systems provide the option to set USB ports OFF or to reduce 5V power supply on the USB ports. However, this may result in faults and aborts in communication.

Therefore please deactivate energy saving functions on USB ports in the device manager of your system.

Connection of your machine is possible at first LPT connector at *ncUSB* or axes controller only.

Warning !

WinPC-NC is a software that controls hardware. The hardware must be properly connected to the PC and configured in WinPC-NC for correct machine operation. The user is responsible for making all connections, performing all safety procedures, and operating the machine.

Care must be taken to correctly connect and test each wire, switch, sensor, or output. Everything should be checked multiple times; having another person review things while you work is a good idea.

Warning !

Care must be take to avoid damage due to static, incorrect signal assignments, and incorrectly inserted plugs/cables/wires.

Do not attempt to set-up the machine until you have familiarized yourself with this manual and the set-up procedures, are rested, and have plenty of time to set-up the machine.

Warning !

It is hightly recommended that the first signal connection is the emergency stop (E-STOP). This way, any unexpected actions can be stopped before damage occurs.

6.2. IEEE 1284 port (LPT) pins
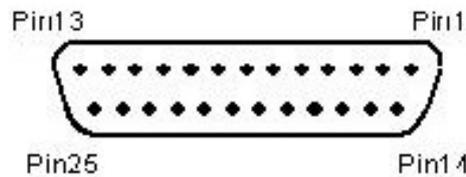
IEEE 1284 parallel port (LPT) pins

Pin assignment LPT1

All IEEE 1284 ports adhere to the 5V TTL standard for inputputs and outputs. LPT1 is available on the *nc100* and *ncUSB* modules. The default pin assignments are:

Pin 1	**OUTPUT**	**spindle on/off**
Pin 2	**OUTPUT**	**direction motor X**
Pin 3	**OUTPUT**	**clock motor X**
Pin 4	**OUTPUT**	**direction motor Y**
Pin 5	**OUTPUT**	**clock motor Y**
Pin 6	**OUTPUT**	**direction motor Z**
Pin 7	**OUTPUT**	**clock motor Z**
Pin 8	**OUTPUT**	**direction motor 4 (i.g.tangential axis)**
Pin 9	**OUTPUT**	**clock motor 4 (i.g. tangential axis)**
Pin 14	**OUTPUT**	**coolant pump on/off**
Pin 16	**OUTPUT**	**current reduction**
Pin 17	**OUTPUT**	**job running**
Pin 10	**INPUT**	**reference switch X**
Pin 11	**INPUT**	**reference switch Y**
Pin 12	**INPUT**	**reference switch Z**
Pin 13	**INPUT**	**sensor/surface block**
Pin 15	**INPUT**	**free**
Pin 18-25		**Ground (0V)**

Pin assignment LPT2

LPT2 is only available on the *ncUSB* module. The default pin assignments are:

Pin 2-9 **OUTPUT** **analog output, 256 binary coded steps**
Pin 16 **OUTPUT** **free**
Pin 17 **OUTPUT** **free**
Pin 14 **OUTPUT** **free**
Pin 10 **INPUT** **free**
Pin 11 **INPUT** **free**
Pin 12 **INPUT** **free**
Pin 13 **INPUT** **free**
Pin 15 **INPUT** **free**
Pin 18-25 **Ground(0V)**

free definable signals to pins

Each IEEE 1284 port (LPT) has pins 10, 11, 12, 13, and 15 available for input and pins 1, 14, 16, and 17 available for output.

WinPC-NC USB :

The Q218 signal (i.e. Spindle Speed/PWM) may only be connected to pins LPT1:14 or LPT1:17.
The Q219 signal (i.e. Toggle/Charge pump) may only be connected to pins LPT1:16 or LPT1:17.

WinPC-NC Professional :

All addon signals can be assigned to any input or output at axes controller.

6.3. Determination of axis resolution

For calculating the required distances and speeds it is necessary to tell **WinPC-NC** precisely the definition of the axis resolution.

axes resolution defined by two parameters

Please open the parameter dialog box parameter - machine and determine the exact data of your mechanics and drives for each axis by the first two parameters.

The parameter axis resolution defines the number of steps or increments per rotation for the corresponding motor. Please consider the electronic settings relating to the macro/micro stepping operation and a possibly integrated reduction ratio.

Motor steps and distance per rotation

The required value is the number of motor steps **WinPC-NC** has to create in order to carry out exactly one rotation round the spindle or the shaft.

The second parameter distance per rotation defines the distance which is made exactly by the number of above specified motor steps. With spindles it is the spindle rise; with belts or gear racks it is the graduated circle size of the pinion.

In our homepage in *Help - HowTo* we present a tutuorial concerning correct settings of axis resolution with new or unknown machines.

Incorrectly set axis resolutions will cause dimensional inaccuracies and inexact speeds.

6.4. Determination of direction

Select JOG MOVE and study the buttons. Each axis is to be **carefully** tested for forwards and backwards motion. Each motion **must** move exactly as expected.

Start with X-axis.

Arrow keys on the right are for motion in the positive direction; on the left for negative motions. Check your machine documentation for where to stand:
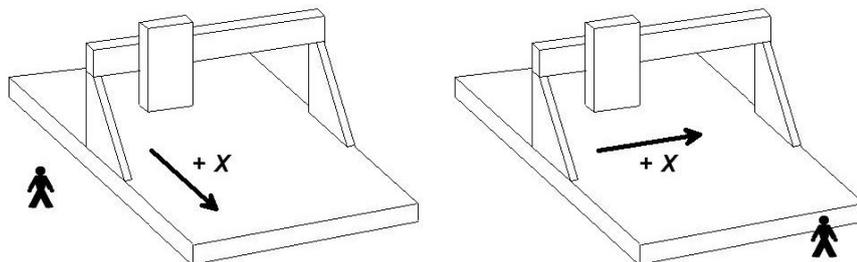
Carefully attempt a positive motion first. If successful, attempt a negative motion.

If the expected motions do not occur, here is a table of common errors, causes, and solutions:

Error	Possible Cause	Solution
Machine does not move	Cable not connected	Connect the cable
	Cable not connected to the correct LPT port	Connect the cable to correct LPT port
	Incorrect pin assignment	Correct the pin assignment
	Machine off	Check if the machine and/or its motion controller are powered on.
Unexpected movement (wrong axis moves)	Incorrect pin assignment	Correct the pin assignment
Axis moves in opposite direction expected	Direction signal works with inverted logic	Change the parameter INVERT MOVE DIRECTION under machine parameters

Once the X-axis appears to be working as expected, repeat the testing procedure on the Y-axis. Correct any issues until the X- and Y- axes appear to be working as expected.

Arrow keys up must move the Y axis back.
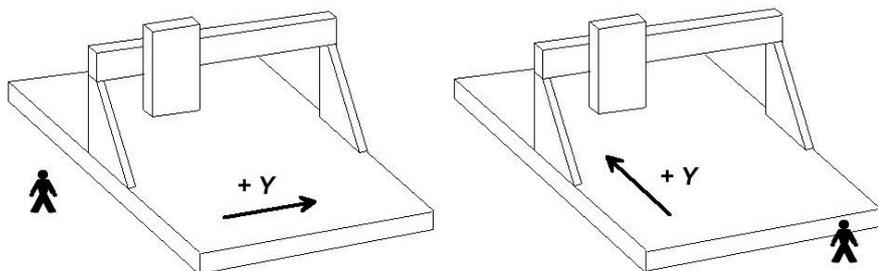
Now test the Z-axis.

The Z-axis moves down when arrow keys showing downwards.
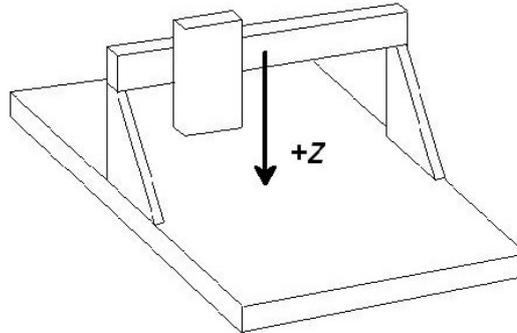
For a direction reversal of one axis it is only necessary to change the parameter INVERT MOVE DIRECTION.

Movement directions are relative to the movement of the tool above a work piece. Movement is opposite that of the arrow keys in order to guarantee the correct movement direction of a tool above the work piece.

6.5. Home and limit switch adjustment

The home and limit switches can easily be checked with the SPE-CIAL FUNCTIONS-SIGNAL TEST.
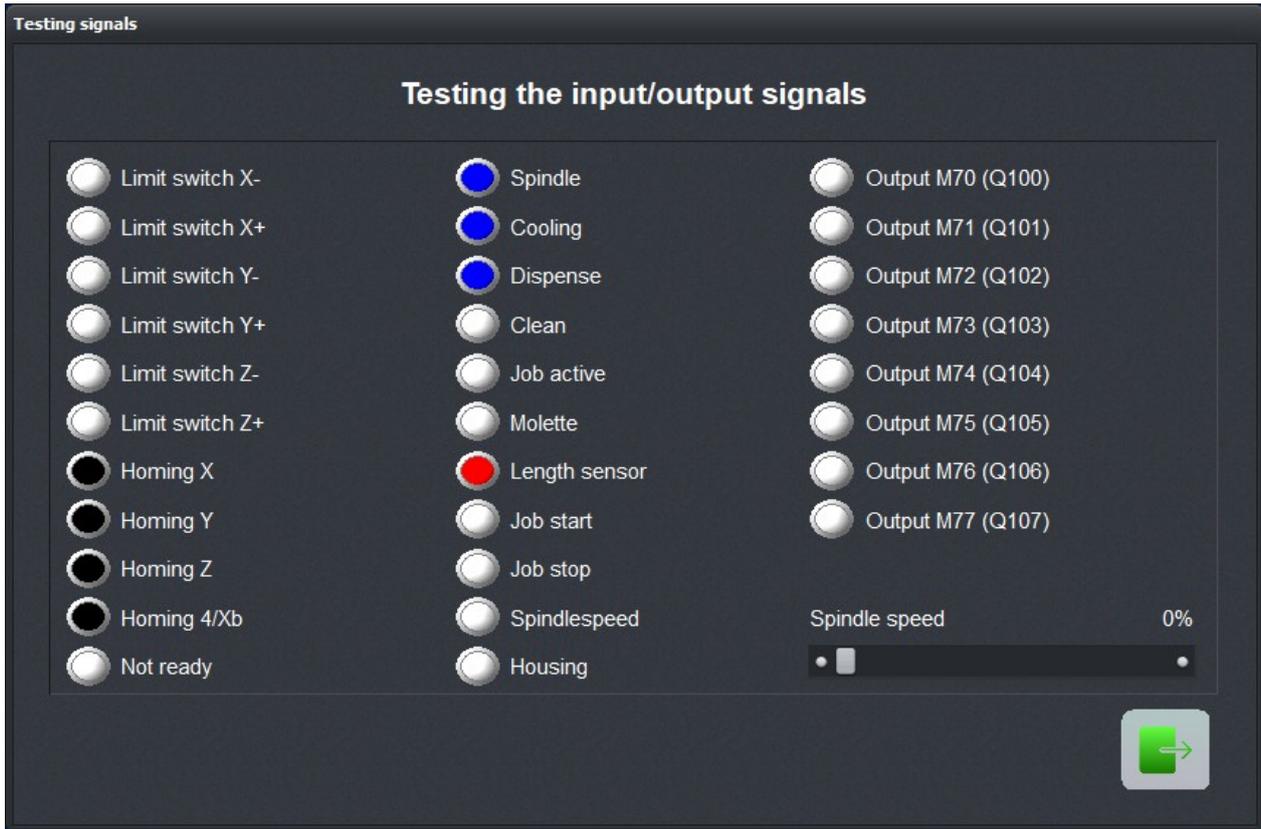
Testing signals

Manually activate switches

Manually activate each switch (home and limit) and check the signal LED in the *Testing signals* window.

The expected LED should be black until it is activated; red when it is activated

Depending on the used switch (NO contact - NC contact) the switching logic may be misapplied, i.e. red colored with the un-pressed switch and black colored with the pressed switch. If this proves to be true, the switching logic of the reference switch must be changed by parameter.

Set switching logic

Please follow the following procedure:

1. Open parameter - basic settings - signal wizard

2. Select the input signal to be defined in upper left window

3. Open dropdown menu and select the connected pin no and assign it to the input signal. If the switch is a NO contact the

assignment must be inverted, i.e. LPT1 Pin10 inv instead of LPT1 Pin10.

4. Test all assignments and control it again with SPECIAL FUNCTIONS-SIGNAL TEST.

A detailed listing of all available input and output signals can be found in next chapter.

6.6. Setup of homing movements

Now that the home and limit switches are installed and work as expected, it's time to let *WinPC-NC* know where they are, how fast they can be located (search and free speeds), and in what order the home switches are located.

Please follow the following procedure:

1. From the Parameters menu, select the BASIC SETTINGS tab and the HOMING tab.

2. Specify the home switch location for each axis of your machine. This will at the positive or negative end of your machine. If you're unsure which is the positive or negative end of your machine, change to JOG MOVE and move your machine in both directions, observing the position counters and noting which direction the axes move as they approach a home switch.

3. Set the appropriate search and free speeds. These should be on the slow side as one does not want to collide with the switches too fast, potentially damaging them.

4. Set the appropriate axis search order. It is usually best to stay with the default (Z-Axis first).

5. Set the machine zero point. It's best to set this at a small distance from the reference position.

6. Start the machine initialization sequence. One axis at a time will move.

7. If one or more axes have their homing switch at positive axis end it might be useful to define the Reference point NOT as a zero point but to the maximum possible distance of this axis.

6.7. Control of settings

Before attempting to initialize the machine for the first time, double check all of your settings. Press function key $\boxed{\text{F8}}$ or select

the corresponding function from the menus, and save your settings.

Activate initialization procedure *for monitoring the* settings

During the initialization procedure, your machine will move one axis at a time. Initially, each axis will move towards its home switch, until it activates. It will then move away from its home switch at a reduced speed until the home switch ceases to activate. This is the reference point for that axis.

If everything has been done correctly, the initialization procedure of your machine should finish without any problems. If an error occurs or your machine moves in an unexpected way, review your settings, make the necessary adjustments, save your parameters, and try again.

Error	Solution
Axis moves in the wrong direction	Define the reference switch at the other end
Axis moves in the correct direction, but very slowly	Switching logic has been adjusted incorrectly and has to be inverted and define the reference switch at the other end
Axis moves slowly in the correct direction, but stops on the switch	Switching logic has been adjusted incorrectly and has to be inverted and define the reference switch at the other end
Axis moves very slowly in the wrong direction	Switching logic has been adjusted incorrectly and has to be inverted

6.8. Additional steps

Once the basic connections and motions have been made and verified, it's time to finish the set-up procedure:

1. Determination the optimal motor ramp and traverse speeds for each axis using SPECIAL FUNCTION-MOTOR TEST.

2. Define these speeds for homing and manual movements.

3. Enable the needed functions and technologies or the available equipment.

4. Connect any remaining input and output signals and verify them via the Signal Wizard.

7. Signal wizard

7.1. Input and output signal management

The management of the input and output signals are fundamental to *WinPC-NC*.

Input and output signals

Input signals make it possible to handle things like:

• Home and limit switch monitoring

• Machine initialization and automatic calibration

• Synchonization with external hardware (e.g. Start).

• Safety device monitoring (e.g. doors, hoods, fences).
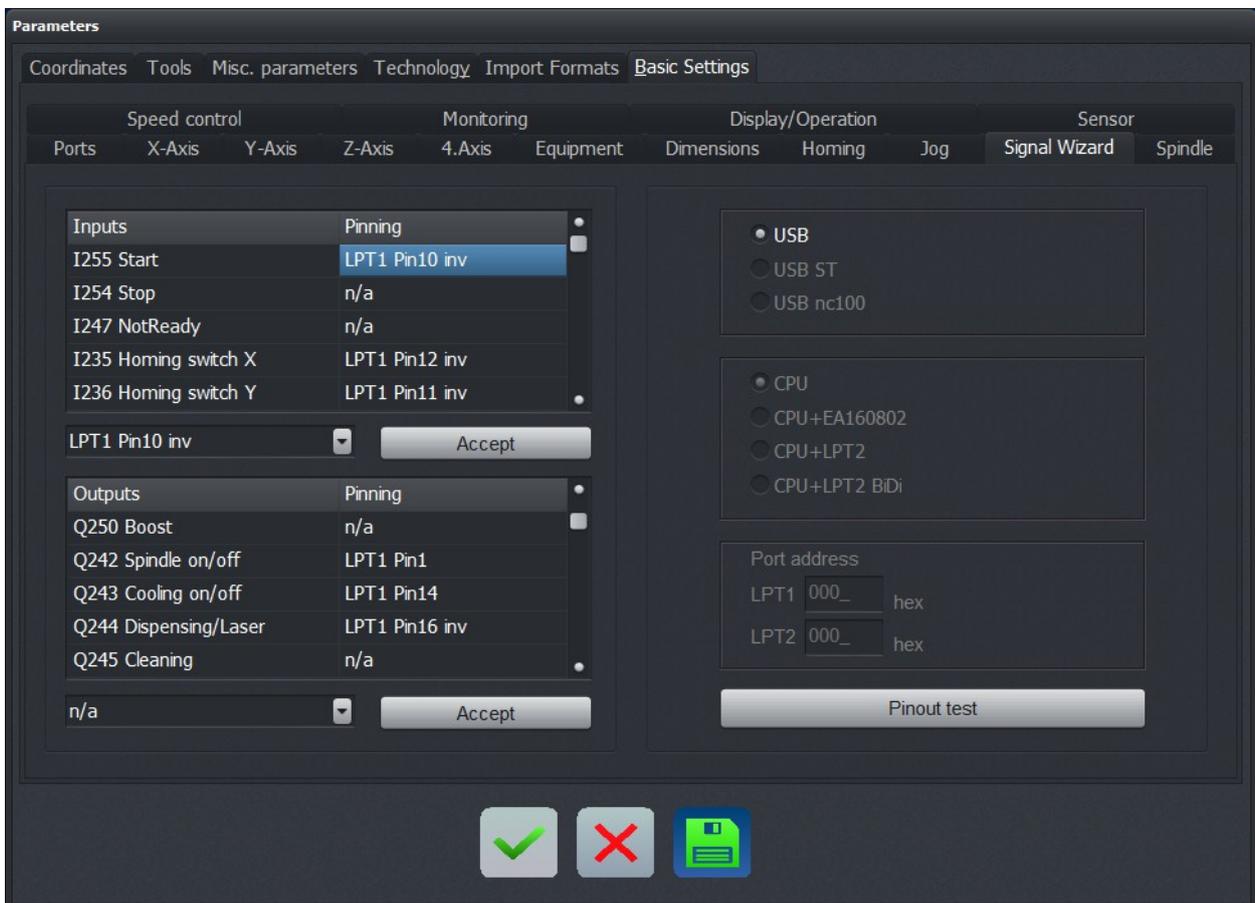
Signal wizard - pin assignment

Output signals make it possible to control things like:

- Spindle power and speed.

- Safety device locks (e.g. doors, hoods, fences).

- Dispensing pump power and speed.

Assignment of the signals

WinPC-NC supports *many* different signals. Only connect and assign those signals that are applicable to your machine and application.
The Signal Wizard makes it is quite simple to assign a signal to any available pin.

Each IEEE 1284 port (LPT) has pins 10, 11, 12, 13, and 15 available for input and pins 1, 14, 16, and 17 available for output.

CAUTION: Please take great care when assigning and testing inputs and outputs. Be vigilant, watching for any unexpected motions or actions.

7.2. Assignment of inputs

Assignment of the input signals

Use the following steps for assigning the input lines:

- Connect a required switch, wire, or sensor to an available pin. Click the *Pinout test* button. The Pinout test window will appear.
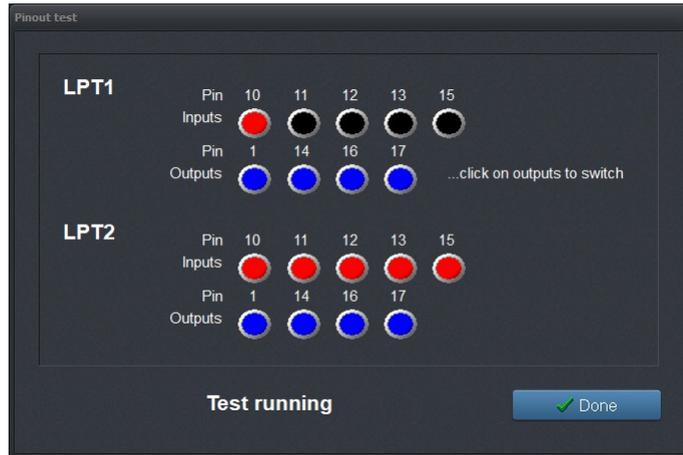
Pin test

- Perform an action which activates the switch, wire, or sensor.

  A normally open (NO) contact is indicated when the LED light changes from black to red when it is activated.

  A normally closed (NC) contact is indicated when the LED light changes from red to black when it is activated.

- Click *Done*. Control now returns to the Signal Wizard.

- Select the required signal from the input list on the left (e.g. I235 reference switch X) and select the pin (e.g. LPT1 Pin11) from the pop up menu next to the *Accept* button.
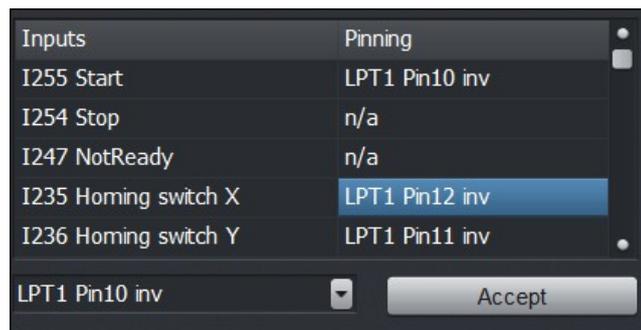
Assignment of the pin num.to the inputs

- With normally open contacts, select the pin number (e.g. LPT1 PIN11). With normally closed contacts, select the inverted pin number (e.g. LPT1 PIN11 Inv).

- Click the Accept button.
  The selected signal and pin number will be displayed in the assignment table.

- Repeat this process for all required inputs

- Save the parameter settings.
  It is a good idea to use SPECIAL FUNCTION-SIGNAL TEST and/or SPECIAL FUNCTION-MOTOR TEST to verify that everything is working properly.

WinPC-NC inputs only react to a HIGH level. Pins that use inverted logic must be assigned properly such that their state generates a HIGH level.

7.3. Assignment of output lines

Assignment of the output signals

The assignment of outputs is similar to the assignment of inputs.

- Connect a required relay, wire, or control to an available pin.

- Click the *Pinout test* button. The Pinout test window will appear.

Testing input and output signals

- Click by mouse pin and observe the level of the signal.
  A blue LED light indicates a LOW level or an output that is switched off. A green LED light indicates a HIGH level or an output that is switched on.

- Click *Done*. Control now returns to the Signal Wizard. Select the required signal from the output list on the left (e.g. Q242 Spindle on/off) and select the pin (e.g. LPT1 Pin1) from the pop up menu next to the *Accept* button.

- Click the *Accept* button.
  The selected signal and pin number will be displayed in the assignment table.
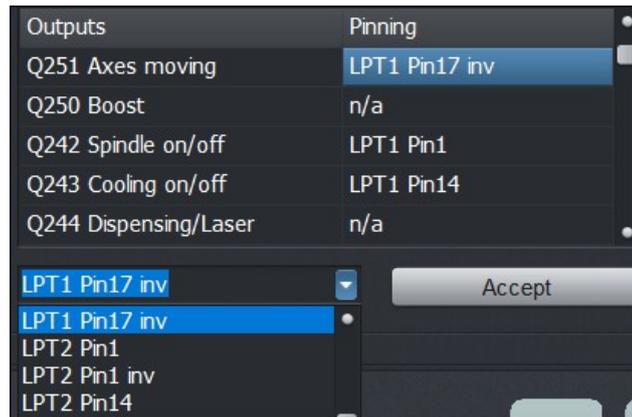
Assignment of pins to output signals

- Repeat this process for all of the required outputs.

- Save the parameter settings.
  It is a good idea to use SPECIAL FUNCTION-SIGNAL TEST and/or SPECIAL FUNCTION-MOTOR TEST to verify that everything is working properly.

7.4. Supported Input Signals

Input signal	Assigned action
I255 Start	Loaded job may begin
I254 Stop	Immediately stops the running job when this signal transitions from LOW to HIGH. After this signal is detected, the machine must be (re-)initialized before further use.
I247 Not ready	Immediately stops the running job when this signal transitions from LOW to HIGH. This is the signal used for the emergency stop (E-Stop). After this signal is detected, the machine must be (re-)initialized before further use.
I235-238 Home switch	Home switch signal for all axes
I239-242 Limit switch-	Limit switch positioned at the negative end of the axis
I243-246 Limit switch+	Limit switch positioned at the positive end of the axis
I234 ZinPositionUp	Synchronization for a semi-automatic drilling job. When this transitions from LOW to HIGH level, it is permissible to move to the next position. ZinPositionUp and ZinPositionDown are mutually exclusive.
I233 ZinPositionDown	Synchronization for a semi-automatic drilling job. When this transitions from LOW to HIGH level, it is permissible to move to the next position. ZinPositionUp and ZinPositionDown are mutually exclusive.
I229-232 Limit switch summary	Only one limit switch signal is available for both ends of an axis. Since these signals are ambiguous, *WinPC-NC* cannot detect the direction of a motion and change direction.
I228 Spindle speed	Spindle has reached its specified speed.

I222 Housing	Safety interlock. Job will not begin until this signal is HIGH.
I221 Length sensor/ Surface probe	Surface or tool length sensor contact acknowledge.
I180 JobSingle	Starts the loaded job in step by step mode; stop after each NC command.
I179 JobStart	The job loaded from the main menu has started.
I178 JobStop	Stop running job.
I177 JobResume	Continue interrupted job.
I100-109 free I160-169 free	Signals available for custom use; macro programming accessible

In *WinPC-NC Professional* the following additional signals are available.

Input signals	Assigned action
I220 Homing switch axis Xb	Reference switches for double X-axis and automatic alignment of gantry
I176 THC Z up	THC plasma height control, Z moving up
I176 THC Z down	THC plasma height control, Z moving down
I199 THC error	THC plasma height control, error
I200 THC Arc good	THC plasma height control, signal for start moving
I201-210 ATC magazin 1-10	Signals for monitoring magazines of automatic tool changer
I181-190 inputs for warnings and messages	These inputs can be used for displaying freely definable messages and for monitoring machine status

7.5. Supported Output Signals

Output signal	Assigned actions
Q255 Ready	The controlling software is ready (HIGH). If an error occurs, the output transitions to LOW.
Q251 Axes moving	Indicates movement (HIGH) or stopped (LOW). This signal can be used to adjust current flow.
Q242 Spindle	Spindle on/off.
Q243 Cooling	Cooling/lubricant on/off.
Q244 Dispensing/Laser	Laser/dispensing on/off.
Q245 Purging with compressed air	An upwards Z-axis motion (HIGH).
Q246 JobRunning	Job is running (HIGH).
Q247 JobEnd	A 100 ms long pulse *(HIGH)* indicating that the job has ended.
Q248 Collet chuck/molette	Controls the automatic chuck in conjunction with an Automatic Tool Changer (ATC)
Q249 Spindle CCW/CW	Spindle rotation direction control (e.g. M04/M03 in G-code NC programs)
Q219 Toggle/ChargePump	Continuous 12.5kHz square wave clock signal.
Q218 Spindle speed/PWM	Pulse Width Modulation (PWM) signal used for spindle speed control.
Q100-115 M code	Maps, respectively, to the M70-M86 codes in G-code programs.
Q221-230 free	Signals available for custom use; macro programming accessible

The following additional output signals are only availablse in
WinPC-NC Professional .

Output signal	Assigned actions
Q217 constant speed	indicates a constant moving speed HIGH, is LOW at ramps or with reduced speed in arcs and corners
Q116 THC ignition	shoots the oxy/plasma burner

8. Additional Information

8.1. Interpreters / import filters

WinPC-NC has import filters for serveral NC data formats :

- HPGL   - MultiCAM 2D/3D
- Gcode (DIN/ISO)   - Isel NCP
- DXF (2D)
- Drilling data (Excellon and Sieb&Meier)
- Postscript (vector information in EPS and AI files)

HPGL Interpreter

data for former pen plotters

The HPGL interpreter recognizes most of the commands in the HPGL plotter language.

Successful tests have been carried out with various CAD and drawing programs:

- AutoCAD®   - AutoSketch®
- Eagle®   - HCAM®
- CADdy®   - Google Sketch®
- QCAD®   - Fusion360
- 2DCAM/3DCAM   - Vcarv
- FreeCAD   - Inkscape
- CorelDRAW

HPGL windows driver

Furthermore, the interpreter can also work with HPGL data generated by the Window HP-Plotter driver.

Alternatively you also can use the driver for plotters *HP7475*, *HP Color Pro* or *HP Draftmaster* .

HPGL syntax

The following commands represent a valid HPGL data file :

PA [*x,y*[,*x,y*]];   Move to absolute position(s)
PR [*x,y*[,*x,y*]];   Move to relative position(s)
PU [*x,y*[,*x,y*]];   Lift and move tool
PD [*x,y*[,*x,y*]];   Lower and move tool
SP *x*;   Select tool
IP *x,y*,[,*x,y*];   Define scaling points
SC *x,y,x,y*;   Define scaling factors
CI *r*[,*a*];   Draw complete circle

AA *x,y,a[,c]*;   Absolute arc
AR *x,y,a[,c]*;   Relative arc

In addition, the interpreter recognizes the following commands although no response is produced on the machine:

DT...;   **LT...;**
LB...;   **VS...;**
IN...;   **CA...;**

The following non-standard extension are also supported:

PP;   Drill hole, lower/lift tool
FCt;   Wait t milliseconds
FC100,t;   Wait t milliseconds
FC101,i;   Wait for HIGH level at input i
FC102,i;   Wait for LOW level at input i
FC200,a;   Set output a to HIGH
FC201,a;   Set output a to LOW
FC300,m; Activates a programmable macro m

The HPGL language prescribes the following syntactical rules, which must be observed when creating or revising the data.

• Coordinates are always pairs of numbers - x, y
• All commands with coordinates must have no, one or more coordinate pairs
• Every command must be followed by a semicolon (;) or a new command
• With PA and PR, the subsequent commands are also set to absolute or relative positioning
• The last command should lift the tool (PU;)

HPGL example

PU;   Start by lifting tool
PA1000,1000;   Then move to 1000/1000
SP1;   Select tool 1
PD;   Lower tool
PA1 000,2000;   Move square
PA2000,2000;
PA2000,1000;
PA1000,1000;
PU;   Lift tool

MultiCAM - Interpreter

HPGL additional commands for handling 3D

The MultiCAM format is an expansion of the HPGL standard that adds 3D capabilities. All HPGL commands are supported however extension for handling 3D may be enabled.

A differentiation is made between 2D and 3D modes. There must always be 3 coordinates in 3D mode (including Z - axis).

ZZ0/1;	Activates 2D or 3D mode
ZU*z*;	Tool lift height of the Z-axis over the zero point (overwrites the safety clearance parameter)
ZD*z*;	Plunge depth (overwrites the depth values in the parameters)
SF*v*;	Feed rate
ZO*d*;	Spindle speed

The syntactical rules of HPGL language apply analogously to MultiCAM.

Drilling data - Interpreter

Drilling data format

The drilling data interpreter recognizes data that follow the Excellon, Sieb&Maier1000, and Sieb&Maier3000 standards.

A drilling file always starts with one or two percent signs and ends with the command M30. In between, there can be any number of lines with X and Y coordinates and tool number T.

Drilling data syntax

The syntax is as follows:

% or %%

[X*number*][Y*number*][T*x*]
.....
M30

There are two different formats. In format 1, the coordinate values are always 5-digit numbers, although it is possible to leave off the trailing zeros (e.g. X021 corresponds to 2100).

In format 2, leading zeros are discarded and the coordinate values therefore comprise different numbers of digits. The drilling

formats prescribe the following syntactical rules, amongst others, which must be observed when creating or revising the data.

- Every command must be in its own line
- If coordinates and a tool command are contained in the same line, then the tool is changed first before drilling takes place at the position
- The coordinates can also include a decimal point, e.g. X123.456
- All lines before the percent sign(s) are regarded as comments and are skipped

Sample of drilling data in format 2

Short drilling program	Comment
%	Start of program
T1	Select tool 1
X1000Y1000	Drill hole at 1000/1000
X1200Y2340	Drill hole at 1200/2340
X2700Y2950T2	Tool 2 hole at 2700/2950
Y1000	Drill hole at 2700/1000
M30	End of program

The same example with format 1 :

Sample of drilling data in format 1

Short drilling program	Comment
%	Start of program
T1	Select tool 1
X01Y01	Drill hole at 1000/1000
X012Y0234	Drill hole at 1200/2340
X027Y0295T2	Tool 2 hole at 2700/2950
Y01	Drill hole at 2700/1000
M30	End of program

G-code interpreter

WinPC-NC also contains a (DIN/ISO) G-code command interpreter. The G-code language is used in professional and industrial applications for programming NC machines.

The following commands are supported:

G-Commands

G00	Linear move at machine rate (rapid)
G01	Linear move at specified rate (machining)
G02	Clockwise circular movement
G03	Counterclockwise circular movement
G04	Dwell time, milliseconds, via F command
G17	Plane XY
G18	Plane XZ

	G19	Plane YZ
	G28	Move of zero point
	G54..59	Zero offset
	G70	Units are Imperial (inches)
	G71	Units are Metric (millimeters)
	G81	Drilling and withdrawal in slow speed Z-depth R-tool lift P-dwell time
	G82	Drilling and withdrawal in rapid speed Z-depth R-tool lift P-dwell time
	G90	Poisitions are absolute
	G91	Positions are relative
	G98	Define subroutines
M-Funktionen	M00	Program stop
	M02	End of program
	M03	Spindle on, clockwise rotation
	M04	Spindle on, counterclockwise rotation
	M05	Spindle off
	M06	Change tool
	M07	Coolant on
	M08	Coolant on
	M09	Coolant off
	M16	Wait for input, input number in F
	M30	End of program
	M66	Change tool, same as M06
	M70..77	Set output 100..107 to HIGH
	M-70..-77	Set output 100..107 to LOW
	M80..87	Set output 108..115 to HIGH
	M-80..-87	Set output 108..115 to LOW
	M90..99	activate macros 1-10
	M106	set output HIGH for 3D printer head
	M107	set output LOW for 3D printer head
Misc commands	N	Line number
	S	Spindle speed in revolutions per minute
	F	Pin number (with M16 command)
	F	Dwell time (with G04 command)
	F	Feed rate (mm/s, mm/min, in/min)
	I,J,K	Circle parameters
	T	Select tool
	X,Y,Z	Coordinates
	U,V,W	Coordinates for 4th axis, if activated
	A,B,C	Coordinates for 4th axis, if activated
	R	Radius of arcs
	R	tool lift for G81/82 drilling
	P	dwell time at G81/82 drilling
	L	Call subroutines (1..20)

Example	%prog2	Start of programm
Square with	N001 G90	Absolute coordinates
rounded	N002 G71 T1 M6	Dimensions in mm, tool 1
corners	N003 G00 X110 Y100 Z10	Speed to 1st position
	N004 G01 Z11	Plunge movement in Z
	N005 G01 X190	Feed movement in straight line
	N006 G03 X200 Y110 J10	Arc about center point
	N007 G01 Y190	etc...
	N008 G03 X190 Y200 I-10	
	N009 G01 X110	
	N010 G03 X100 Y190 J-10	
	N011 G01 Y110	
	N012 G03 X110 Y100 I10	
	N013 G01 Z10	
	N014 G00 X0 Y0 Z0	Rapid speed to zero point
	N015 M30	End of program

Bear in mind the following points when writing G-code programs:

- The G-code program file must be a text file, not a word processing file.

- The first line should be a percent sign (%) followed by the name of the program.

- At least one tool has to be selected e.g. with T1 M6 in the program head (M6 is absolutely necessary)

- The feed rate units are set in PARAMETERS-UNITS

- The command number can be defined using N commands

- The arc commands (G02/G04) support both the I, J, K (position) and R (radius) variants. They cannot be mixed. Positive radii produce an arc less than 180° and negative radii an arc greater than 180°.

- Up to 20 subroutines may be defined after M30 (the end of program). A subroutine begins with G98 Lx and ends with G98 L0. A subroutine call made by invoking Lx.

- G54…G59 define job zero points. A definition of the zero point combines the position data on the line with the current zero point.

- Activate one of these zero points by usings its number (e.g. G54) on a line by itself (no other parameters).

Example drilling cycle as subroutine	%prog2	Start of the program
	N001 G90	Absolute coordinates
	N002 G00 X110 Y100	Rapid traverse to 1st position
	N003 L1	Call drilling cycle
	N004 G00 Y110	Next position
	N005 L1	etc...
	N006 G00 Y120	
	N007 L1	
	N008 G00 X0Y0	
	N009 M30	End of program
	N020 G98 L1	Define drilling cycle
	N021 G01 Z20	
	N022 G00 Z0	
	N023 G01 Z25	
	N024 G00 Z0	
	N025 G01 Z30	
	N026 G00 Z0	
	N027 G98 L0	End of subroutine

ISEL NCP Interpreter

The ISEL NCP format is largely associated with companies in the ISEL group (e.g. ISEL, MES). *WinPC-NC* is able to interpret NCP programs - excepting the track data and the track sections.

easy substitution for older ISEL machines

It is very easy to continue your work from an ISEL machine at a new and different machine using *WinPC-NC*.

The commands supported by this interpreter are:

FASTVEL	**VEL**
MOVEABS	**FASTABS**
MOVEREL	**FASTREL**
PLANE	**CCWABS**
CWABS	**CCWREL**
CWREL	**WAIT**
GETTOOL	**HALT**
SPINDLE	**COOLANT**
DRILLDEF	**DRILL**
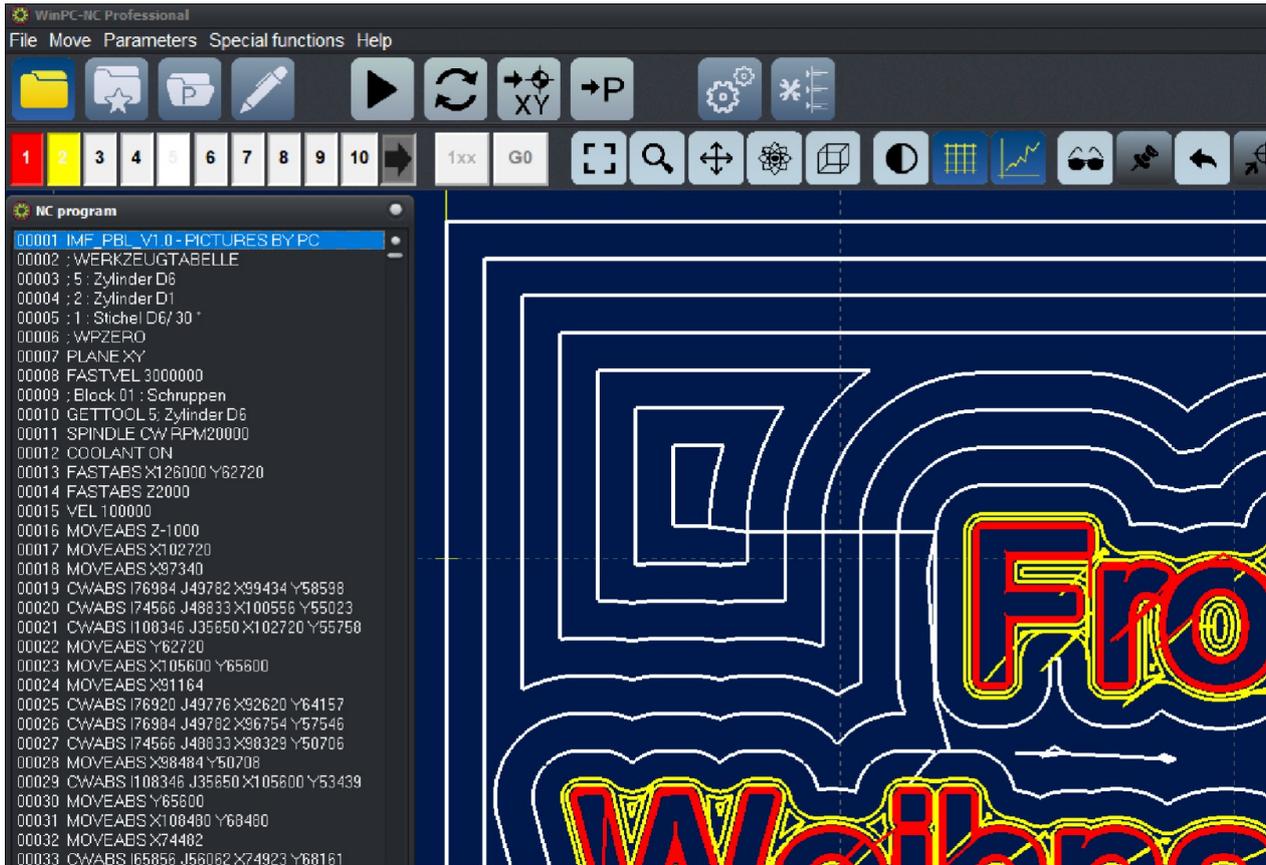
ISEL NCP program - Graphic and program display

Postscript Interpreter

WinPC-NC is able to interpret vector data contained in Postscript files generated by many graphics programs (e.g. Adobe Illustrator®, Corel Draw®). All other data will be discarded as they have no affect on NC processing.

Always create PS files without a header.

A realtime display of the program or postscript files is not available.

Please restrict EPS/AI files to version 3 or 8.

DXF import

WinPC-NC can import files in 2D DXF format. DXF is a widely used data exchange format.

Please create your DXF files in version R10 or R12 or version R14.

WinPC-NC can import most elements from DXF files like LINE, POLYLINE, SPLINE, BLOCKS, VERTEX, ARC, CIRCLE, INSERT, POINT and more.

8.2. Error messages

Error messages are displayed in a small area just below the *Display Area*. Messages in a gray box are status messages, no fault is indicated.

The following error messages may occur:

- **Limit switch reached**
  The machine moved until one of the limit switches tripped. The calibration of the machine is now in question. When this occurs, jog away from the offending limit switch and reinitialize the machine.

- **HPGL Syntax error**
  There is an invalid HPGL command in the plotter file.

- **Connection canceled**
  The connection to the DLL communications module has been interrupted. This happens when other Windows programs or drivers interfere with real-time tasks.

  If this message appears, ensure that all Windows power saving features have been deactivated, including those used for USB ports.

- **Incorrect parameter**
  A WPI or WPW file has been corrupted. One or more values are unacceptable.

- **Stop signal found**
  The stop signal has been detected. Carefully determine what has occurred and rectify the problem.

- **Machine has not been initialized**
  WinPC-NC and all related hardware must be initialized before any machine operations can occur. Failure to do so can result in machine, spindle, or tool damage.

- **Error xx from controller**
  Communication between *WinPC-NC* and one of its DLL modules has failed. Please report this error to the program authors and providing as many details as possible.

- **Data cannot be displayed**
  Graphical display of the current NC data is not possible. The data may be corrupted, not be in a format that *WinPC-NC* understands, or in a format that is currently disabled.

- **Tool not yet defined**
  The NC data is referring to a tool number that has no data specified for it. The appropriate parameters must be defined prior to use.

- **Required tool not defined**
  The NC data is referring to a tool number that has no data specified for it. A plunge depth and speed must be defined for every tool; HPGL files require a feed speed to be defined.

- **No tool file found**
  No tool file can be found. *WinPC-NC* will continue using its default values.

- **Machine is positioned on a limit switch**
  An attempt was made to initialize the machine when the machine is already positioned at one or more of the limit switches. Jog the machine to a position that is well away from all of the limit and home switches and then initialize the machine.

- **Machine positioned on a home switch**
  An attempt was made to initialize the machine when the machine is already positioned at one or more of the home switches. Jog the machine to a position that is well away from all of the home and limit switches and then initialize the machine.

- **Command rejected by controller**
  An internal error occurred when *WinPC-NC* was communication with a DLL module. Please report this error to the program authors and provide as many details as possible.

- **Machine not ready**
  The machine has not been switched on or a fault is present. Carefully check the machine and its interfaces and resolve the issue.

- **USB communication failed / Timeout**
  WinPC-NC was unable to communicate with the external USB module.
  Check the following:
  - Check all the cabling.
  - Only the cable provided with the USB module should be used.
  - No USB hubs or switches should be interposed between the PC and
    USB module.
  - Ensure that all Windows power saving features have been deactivated,
    including those used for USB ports

- **Housing not closed**
  A protective hood or a similar protective device has not been closed; no
  process can be started until the hood is closed.

- **File with invalid format**
  The selected NC file contains data that does not match the selected NC
  data format. Change the NC data selection to match that of the file.

- **Help file not found**
  The *WinPC-NC* help files cannot be located. They are collected in PDF
  directory.

- **Work piece exceeds the size of the working area**
  One or more parts of the work piece extends beyond the boundaries of
  your machine.

  See if it is possible to relocate the work piece to avoid extending beyond
  one or more of the boundaries. Pay particular attention to the Z-axis - the
  work piece, tool, and spindle must fit with the Z-axis limitations of your
  machine.

- **Next position outside working area**
  While the current position is within the boundaries of your machine, the
  next NC command is requesting a move that is outside the boundaries.
  See if it is possible to relocate the work piece to avoid extending beyond
  one or more of the boundaries.

- **Unknown command**
  The NC file contains an unknown command. Processing ceases. The job
  can be cancelled or continued from command immediately following the
  unknown command.

- **Cannot create file**
  WinPC-NC tried, unsuccessfully, to create a file. The selected location is
  on a write protected drive, the drive is full, or you lack sufficient access
  rights. Correct the problem or switch to another location.

- **No park position defined**
  Moving to the park position is only possible if it was previously defined.
  Define a park position.

- **Tool moves in too far**
  The Z-axis move would cause the machine to exceed the defined maximum depth - or a Z-axis depth alarm has been raised.

- **G-code or DIN/ISO syntax error**
  There is a syntax error in the G-code file being processed. Please edit the file, repair this issue (which may occur more than once), and restart the working process.

- **Out of memory**
  WinPC-NC cannot process any further data as it has exhausted all of the memory available to it. Please consider adding more RAM or extending Windows virtual memory. *WinPC-NC* **must be restarted**.

- **Incorrect realtime-DLL**
  There are communcation problems between the actual realtime module and *WinPC-NC* which are probably caused by another license.

- **DIN-syntax error / DIN-circular arc error-**
  There is an invalid command at the indicated line of the DIN/ISO file; The circular arc specification does not match the indicated center point.

- **Error in speed**
  The defined speed does not match the corresponding machine parameters.

- **Keypad error**
  A specified key pad cannot be located. Check the cabling.

- **WinPC-NC does not run with this DLL**
  The files WINPCNC.EXE and WCNCCON.DLL are not compatible with each other or are not issued under the same license. Please re-install and activate the program again.

- **Profile- file incorrect / incompatible**
  The selected profile file is incorrect, corrupted, or incompatible with the license for the version of *WinPC-NC* currently running.

- **Failure in loading profile**
  An error occurred while loading the selected profile. Please check all parameters and settings prior to further use of the system.

- **G00/01 after G54 only one axis**
  The zero point setting in G-code or DIN/ISO programs using G54-G59 commands requires a command to move and continuation has to be effected by all axes.

- **3D Space Mouse not found / not installed**
  A 3D Space Mouse is configured but it could not be found. Please check drivers and cables.

- **Licence file not found or invalid**
  Each *WinPC-NC* licence needs a unique licence file called WINPCNC.LIC. This file could not be found or is invalid or does not match to the connected hardware.

- **USB hardware not found/invalid**
  WinPC-NC USB could not establish a safe communication to the USB hardware module. Please check connection and try a different USB port.

- **Machine is on homing switch**
  You try to initialise the machine but at least one of the homing switches is active and pressed. If you use the same pin no for all switches a reference move only can be started if all switches are free.

8.3. Special and OEM versions of *WinPC-NC*

Customized solutions on request

We always appreciate ideas and proposals.

If desired, we are gladly prepared to pass on our technical know-how in your projects, to provide specific and customized solutions concerning any hard- or software requirements or arrange for best integrating our standard products. Please do not hesitate to contact us without any obligation.

A number of specific and customized solutions, combined with special functions or input options, could be realized in the past by using our well-engineered solution system *WinPC-NC Professional*, e. g.best servicing of the machine and using additional components and features for creating and controlling.

Some examples :

- Dosing plants for glass inserts and reconstructed glasses with motifs

- Laser welding machines and laser marking devices

- Plasma and flame cutting machines

- Laboratory applications for complex measuring tasks

- Machines with automatic changeable double heads

- Welding plants and friction welding plants

- Robot- and automation solutions

- and many other applications besides…

Using the axis controller of **WinPC-NC Professional** you have also the possibility for direct control of external programs. The communication protocol of the axis controller is carefully documented and can be provided on request. Thus any other host systems can be used, independent of the **WinPC-NC** user interface.

Please contact us for special requirements. We are sure to be able to provide an adequate solution or be of assistance in realizing your project – with or without *WinPC-NC*.